The STARRETT BOOK for MOTOR MACHINISTS and AUTO REPAIRMEN

STARRETT BOOKS
VOLUME III

THE STARRETT BOOK

for

MOTOR MACHINISTS
and AUTO REPAIR MEN

Volume III of The Starrett Books

PRICE 75 CENTS

THE L. S. STARRETT COMPANY

World's Greatest Toolmakers

ATHOL, MASSACHUSETTS

PREFACE

In presenting Volume III of The Starrett Books it has been our aim to provide information for the Motor Machinist and Auto Repair Man that is comparable with that offered to the Apprentice and Journeyman Machinist by Volumes I and II.

In its preparation we have made no attempt to deal with specific motor troubles or the peculiarities of particular makes or models of cars, nor have we included any shop "kinks", hints on "trouble shooting", etc. On the contrary, we have confined the text of Volume III to such subjects as would aid the man in the service station or repair shop in gaining a better understanding of the uses, methods of operation and the value of machine and precision tools.

While individual credit has been given wherever possible, a very large portion of the text of the book represents the combined opinions and experiences of a considerable number of men, each of whom is an authority in his field. This being the case, a blanket acknowledgment of our indebtedness to the publishers of the various automobile papers, to their editorial staffs, and to the contributors to their columns, appears to be the only practical solution. We especially wish, however, to express our appreciation of the assistance so generously given by American Automobile Digest, Automotive Industries, Automotive Merchandising, Motor Service, Motor Age and Motor World.

THE L. S. STARRETT CO.

Athol, Mass.

This is, in a sense, a companion volume to Volumes I and II of The Starrett Books — The Starrett Book for Machinists' Apprentices and The Starrett Data Book for Machinists — digests of which will be found at the end of this book.

THE STARRETT BOOK

FOR MOTOR MACHINISTS

MEASURING TOOLS*

In the automobile shop all measurements required, with a few exceptions, are lineal or length. In some few cases it is necessary to know or ascertain area, which is length multiplied by breadth, while in still fewer instances volume must be figured, which is length times breadth times height. As all these measurements are primarily secured by multiplying distances, the actual shop methods of measuring distances—the precision tools used—are extremely important.

In the United States, automobile measurements are all in inches. Wheelbases, tire sizes, cylinder diameters, shaft diameters, etc., are all expressed as so many inches and fractions or decimals of an inch. The foot and the yard are not used. The standard inch is one thirty-sixth of the standard British yard. The United States government owns two exact copies of the British standard yard, these copies being made of "invar" metal which has a minimum of expansion and contraction under atmospheric temperature changes. The L. S. Starrett Company has, in turn, exact copies of the United States copies so that very exact instruments can be constructed to measure down to one ten-thousandth part of an inch. To illustrate what a very tiny distance .0001 is, it is only necessary to know that a safety razor blade is .006 in. thick, paper commonly used for printing books is about .002 in. and a human hair varies from .002 to .005 in.

In France and in many parts of Europe, the unit of measurement is the meter, which is 39.37 U. S. or British standard

*See also pp. 13–28, Vol. I, Starrett Books, and pp. 159–168, Vol. II, Starrett Books.

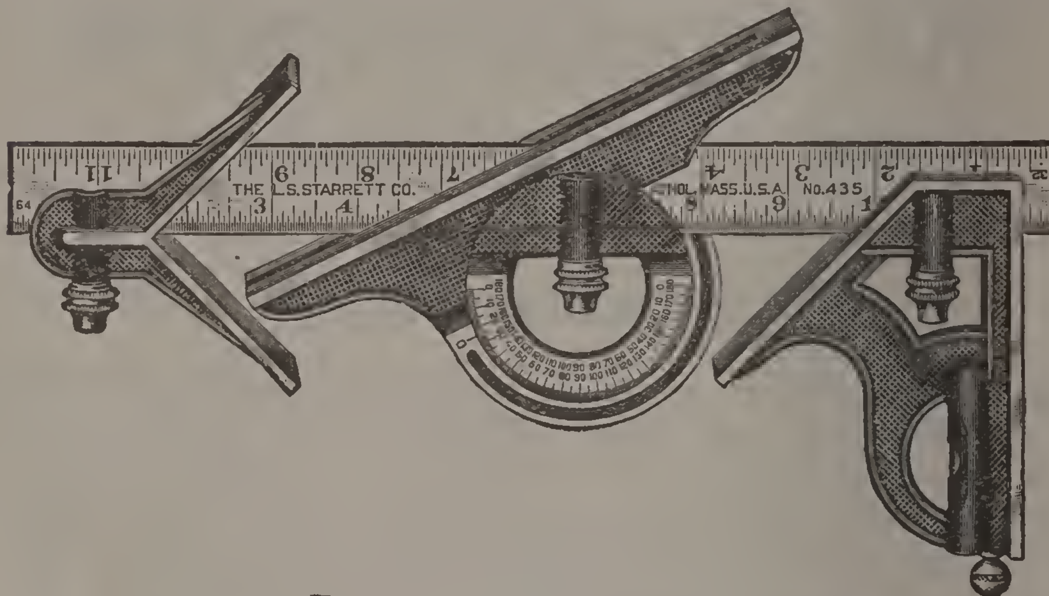


Fig. 1—Combination Set

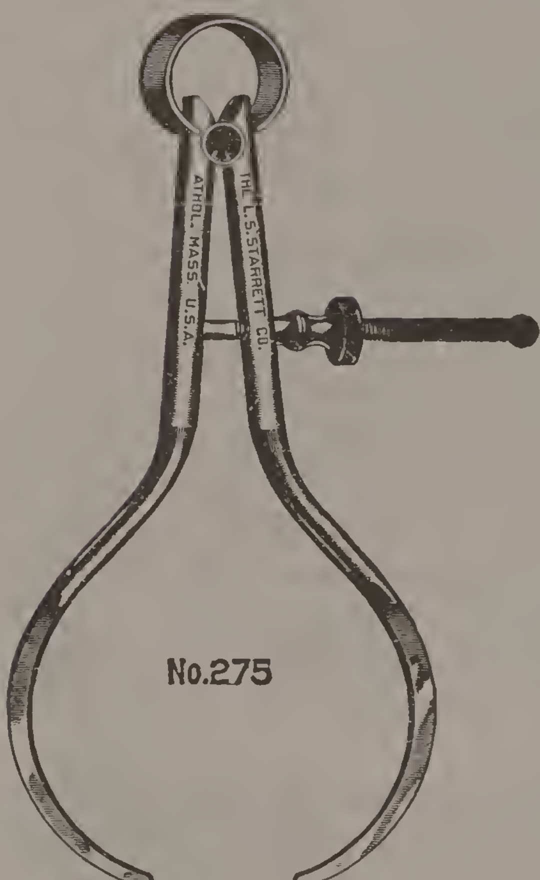


Fig. 2—Toolmaker's Calipers

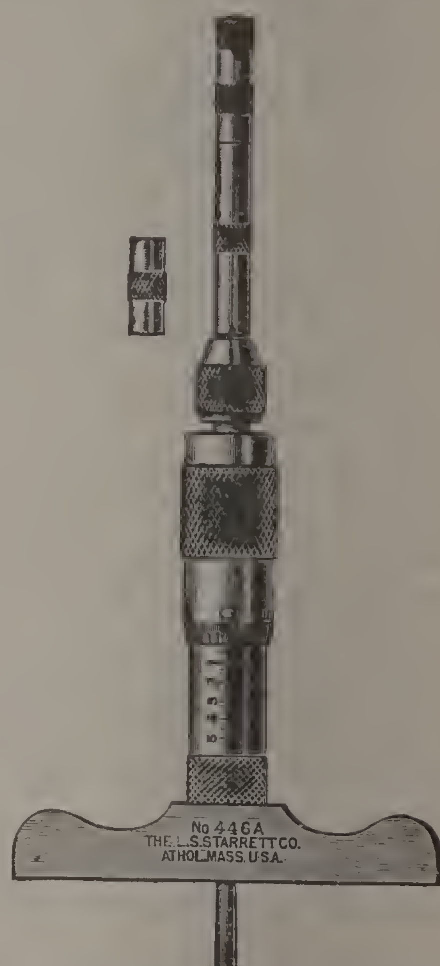


Fig. 3—Micrometer Depth Gage

inches in length. The sub-divisions of the meter are the centimeter which is 1/100 of a meter and the millimeter which is 1/10 of a centimeter.* The only automobiles using metric measurements in the United States are those imported from other countries. There are, however, some American-made cars using metric threads on spark plugs.

Most automobile parts are manufactured to very exact sizes. In some cases the parts must be within .0005 in. of the correct size. Piston pins, pistons, cylinders, valves, king pins, crankshafts and camshafts are generally made to within .001 or .002 in. tolerance.

Anti-friction bearings, both ball and roller types, are often finished to within .0005 in. or half a thousandth of an inch.†

It can be seen, therefore, that good automobile work requires working to very close limits, much closer, in fact, than in the general run of machine shop practice. Dial gages and micrometers form an important part of modern shop equipment and the shop mechanic should know how to operate these as well as inside and outside calipers, protractors, straight edges, steel squares, dividers, steel rules, surface plates, surface gages, center punches, scribers and slide calipers and other precision tools.

FLAT WORK

In general, the worker on flat work will need to be provided with steel rules, dividers, protractors, straight edges, steel squares, surface, height, depth and thickness gages, center punches, parallels, slide calipers, etc.

ROUND WORK

For round work the measurements are by contact, and the usual tools are those having contact points. Contact measurements are made in two ways: (a) The contact tool

*For conversion of millimeters into decimals of an inch see p. 161, Vol. II, Starrett Books.

†For further information on tolerances see pp. 32–33, Vol. I, Starrett Books, and pp. 16–17 and 152, 153 and 154, Vol. II, Starrett Books.

is first set to some standard of length, as, for example, a steel rule, or a standard gage. The "set" dimension may then be used as a standard for testing the work. (b) The reverse of this method may be used for determining sizes, viz.: by first setting the contact points to the surfaces of the work, afterward using the steel rule or standard gage to read the size.

"FEEL"

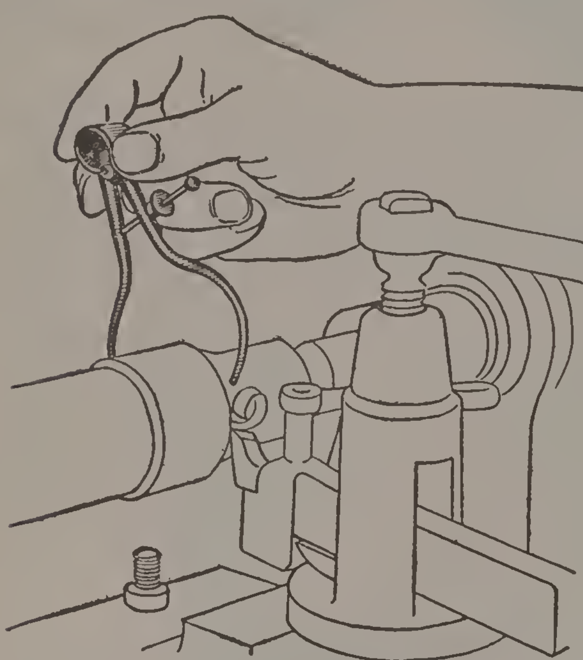


Fig. 4—Calipering Round Work on Lathe

The accuracy of all contact measurements is dependent upon the sense of touch (feel). In the case of skilled workmen, as for example toolmakers, the sense of touch is highly developed. Using suitable contact measuring tools, the skilled mechanic can readily "feel" the difference in contact made by changes of dimensions as small as .00025 in.

In the human hand the sense of touch is most prominent in the finger-tips. Therefore, the contact measuring tool should be held by the fingers only, and in such a way as to bring it in contact with the finger-tips. If the tool is harshly grasped by the fingers, the sense of touch or "feel" is much reduced. For this reason the tool should be delicately and lightly held instead of gripped tightly.

While it is possible to transfer by "feel" a length with an error not exceeding one-quarter of one-thousandth inch, the results are not always easily read. In fact, the personal equation is so great that most mechanics prefer to use direct-reading tools for all accurate contact work.

To illustrate, a "fit" which an apprentice—depending on "feel" alone—would declare perfect, would be instantly rejected by an experienced machinist using the same method of testing and would be shown to be considerably "out" if checked with proper precision measuring tools. Again, the question of "feel" plays a great part in determining the accuracy of work even when precision tools are used. Scarcely any two men will "set" a micrometer or a pair of calipers exactly the same on a piece of work. A comparatively inexperienced man is apt to literally jam the spindle of a "mike" on the work—or else go to the other extreme and not get sufficient contact—simply because his sense of "feel" is not sufficiently developed. An experienced man—having a good sense of "feel"—doing the same work and using micrometers measuring by thousandths of an inch, will turn out work in which the actual variance is less than a ten-thousandth of an inch. The more common tools for contact measurements are inside and outside calipers, used in conjunction with steel rules, plug and ring gages, and dimension blocks. These, however, are not direct-reading tools. The direct reading tools in common use are the caliper square and the micrometer caliper.

CALIPER SQUARES

Caliper Squares—and in this type are included Slide Calipers, Slide Rule Calipers and Circumference Gages, Micrometer Caliper Squares, Vernier Calipers and Vernier Height Gages—are fundamentally combinations of contact points and graduated steel rules. In all of these tools one contact is generally fixed and the other adjustable. When in use the fixed contact is placed against one surface of the object to be measured and the adjustable contact brought up against the other surface. The distance between the contact points may be read direct from the scale on the beam of the tool. Caliper Squares, Slide Calipers and Micrometer Caliper Squares are not commonly used in Automobile repair work, their place being taken by the Micrometer Caliper, which

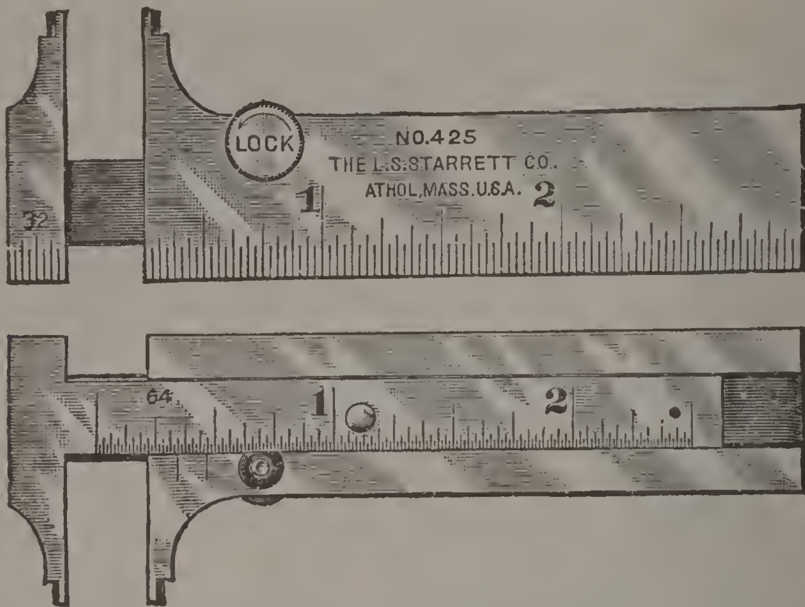


Fig. 5—Caliper Squares

is both more accurate and more convenient. The use of the Vernier Caliper and Vernier Height and Depth Gages is most commonly restricted to the more difficult machine tool operations, though many expert machinists, employed in automobile repair work, find them great savers of time as well as productive of more accurate work.

VERNIER CALIPERS

The Vernier Caliper is a refinement of the Caliper Square and permits the taking of measurements in thousandths of an inch. Apart from the Vernier attachment—the construction and use of which is explained in a later paragraph—the Vernier Caliper is substantially the same as a Micrometer Caliper Square and is used in the same manner.

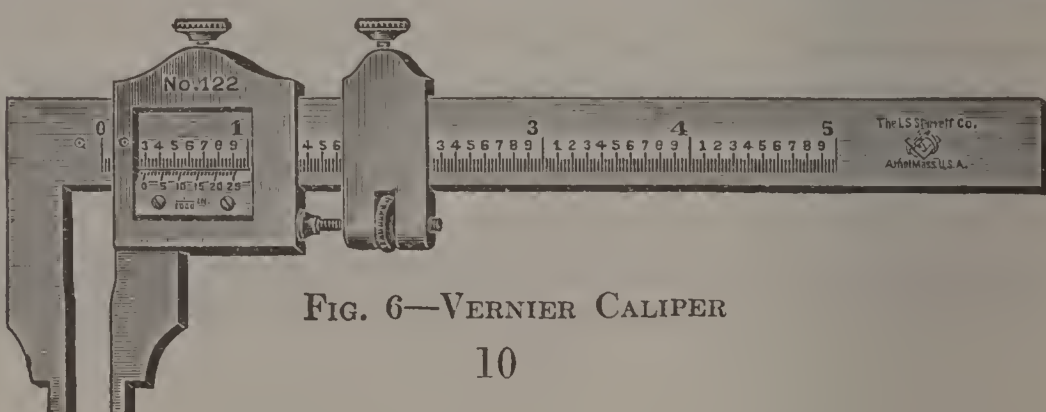


Fig. 6—Vernier Caliper

VERNIER HEIGHT GAGE

Another adaptation of the vernier is the height gage. By means of the vernier it is easy to make readings as minute as one-thousandth part of an inch. This instrument is used chiefly where close, accurate measurements of height must be obtained.

By means of suitable adjustments, one of which is shown on the accompanying illustration (C attachment), its use is extended to include making accurate measurements of depth.

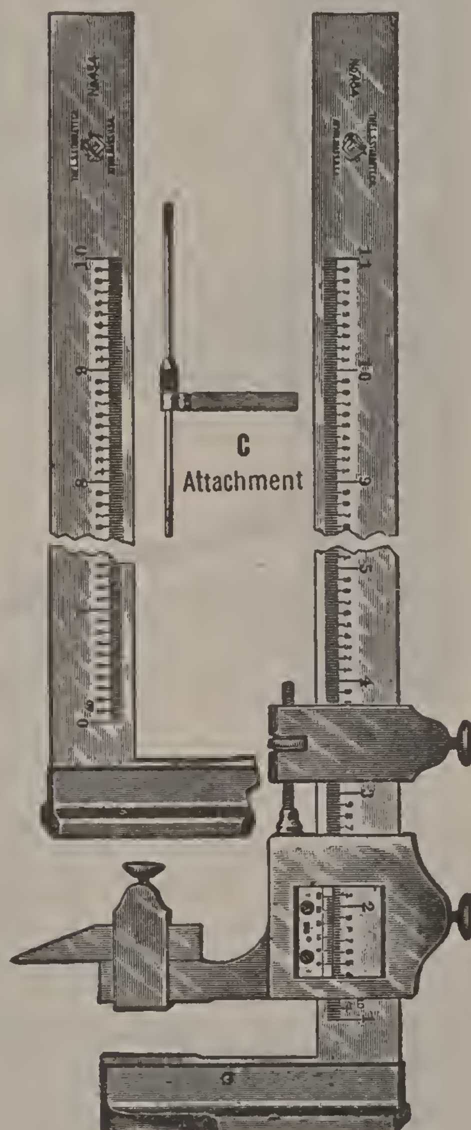


Fig. 7—Vernier Height Gage

MICROMETER CALIPERS

The limit of accuracy obtained by measuring between contacts depends on the graduations on the instrument. It is evident that as the fineness of the graduation increases, the chances for mistaking one graduation for another also increase so that some other method of determining extremely accurate measurements must be used.

Today, the common instrument for making measurements finer than 1/100 of an inch is the micrometer caliper. This tool combines the double contact of the slide calipers with a micrometer screw adjustment which may be read with great accuracy, as may be understood when it is realized that threaded spindles with a limit of error of .001 in. in

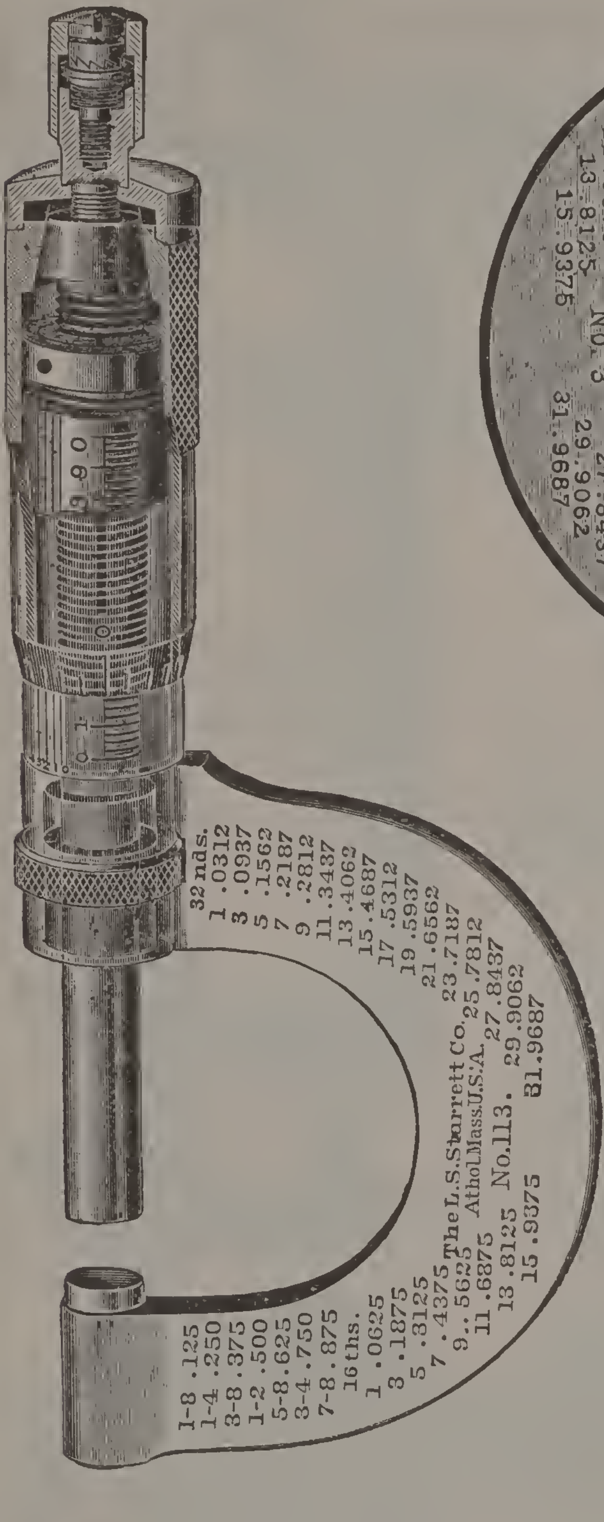


Fig. 8—Phantom View of Outside Micrometer

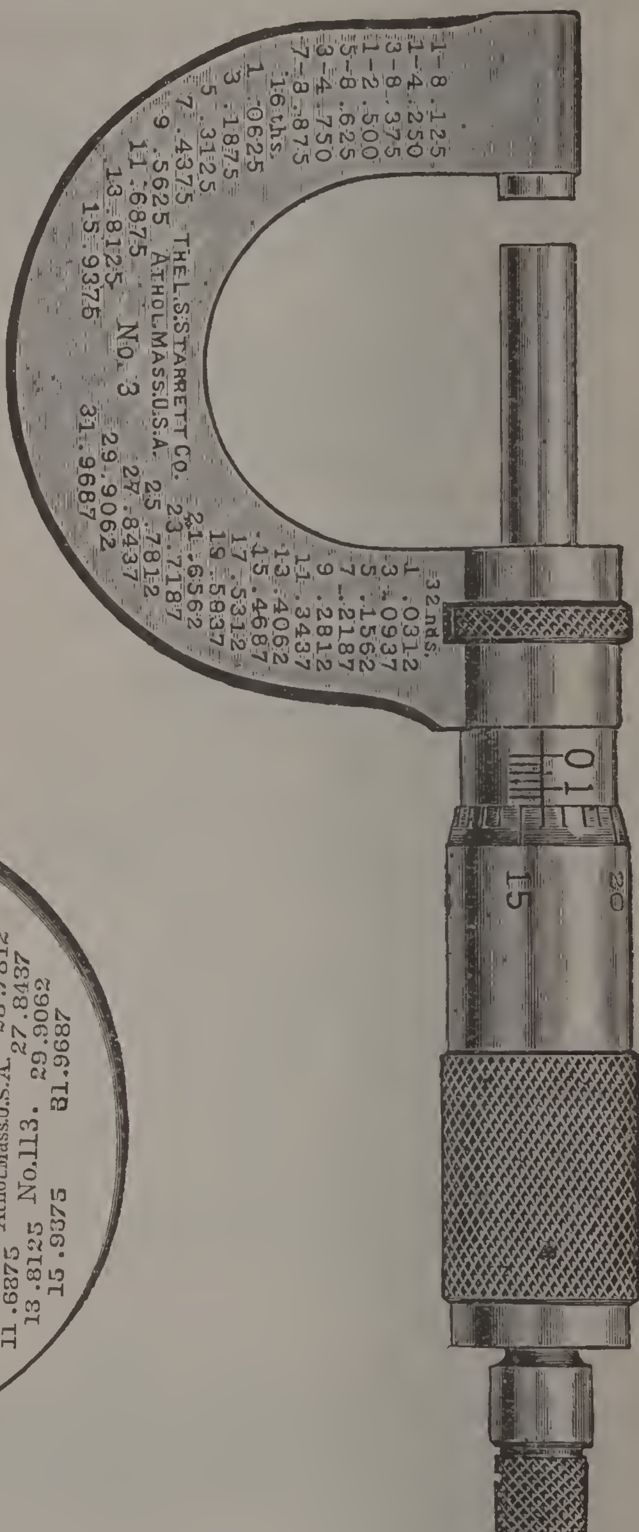


Fig. 9—Outside Micrometer

one-foot lengths are commercially possible. In micrometer construction, where used length of screw thread is only one inch, the degree of error is negligible.

A micrometer head consists of a spindle, forty threads to the inch, fitted through a threaded sleeve, having an enclosing thimble fastened to its outer end. Suitable graduations made axially on the adjustable sleeve, combined with the graduations on the edge of the rotating thimble, give direct readings of one-thousandth part of one inch. By using a vernier scale on the sleeve, direct contact readings as small as one ten-thousandth part of one inch can be readily made.

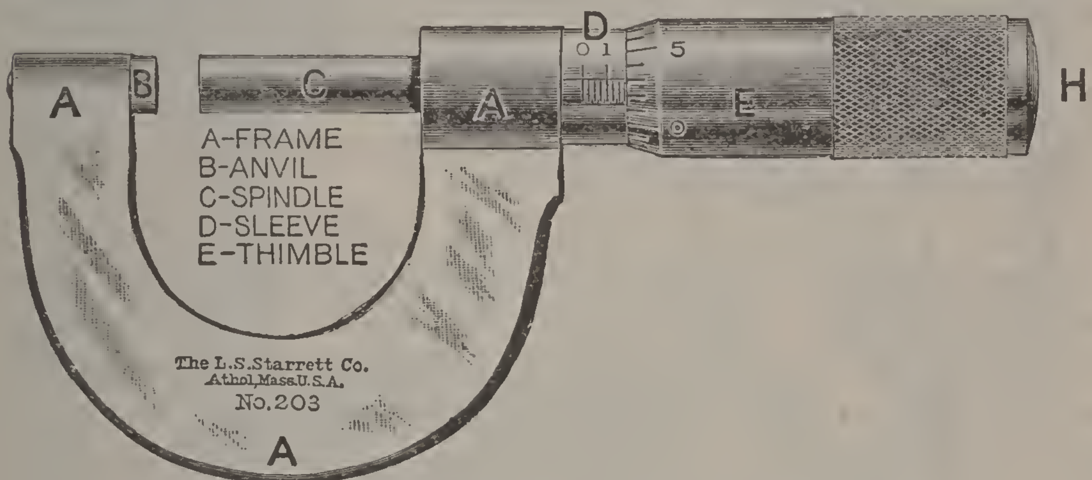


Fig. 10—Parts of a Micrometer

Micrometer screws are mounted in a frame which may be varied in shape and size according to the type of work on which the tool is to be used. The contact points are also shaped to the particular use for which the tool is designed. In general, however, micrometer points are ground and lapped parallel to each other. Micrometers for either inside or outside measurements are purchasable in a variety of styles and of the highest degree of accuracy, convenience and finish.

HOW TO READ A MICROMETER

The pitch of the screw thread on the concealed part of the spindle C (See Fig. 10) is forty to an inch. One complete

revolution of the spindle, therefore, moves it lengthwise one-fortieth (or twenty-five thousandths) of an inch. The sleeve D is marked with forty lines to the inch, corresponding to the number of threads on the spindle. Each vertical line indicates a distance of one-fortieth of an inch. Every fourth line is made longer than the others, and is numbered 0, 1, 2, 3, etc. Each numbered line indicates a distance of four times one-fortieth of an inch, or one-tenth.

The beveled edge of the thimble E is marked in twenty-five divisions, and every fifth line is numbered from 0 to 25. Rotating the thimble from one of these marks to the next, moves the spindle longitudinally one twenty-fifth of twenty-five thousandths, or one-thousandth of an inch. Rotating it two divisions indicates two-thousandths, etc. Twenty-five divisions will indicate a complete revolution, .025 or one-fortieth of an inch.

To read the micrometer, therefore, multiply the number of vertical divisions visible on the sleeve by twenty-five, and add the number of divisions on the bevel of the thimble, from 0 to the line which coincides with the horizontal line on the sleeve. For example, in the engraving, there are seven divisions visible on the sleeve. Multiply this number by twenty-five, and add the number of divisions shown on the bevel of the thimble, 3. The micrometer is open one hundred and seventy-eight thousandths. (7 x 25 equals 175 and 175 plus 3 equals 178.)

HOW TO READ A VERNIER

Readings in thousandths and ten-thousandths of an inch on caliper squares, micrometers, etc., are obtained by the use of a Vernier, named after a Frenchman, Pierre Vernier, who invented the device in 1631. For the Vernier caliper, the scale on the tool is graduated in fortieths of an inch (.025). On the Vernier plate (See Fig. 11) is a distance divided into twenty-five parts, and these twenty-five divisions occupy the same distance as twenty-four divisions on the scale.

The difference between one of the twenty-five spaces and one of the twenty-four spaces is one twenty-fifth of one-fortieth, or one-thousandth of an inch.

To read the tool, note how many inches, tenths (or .100), and fortieths (or .025) the 0 mark on the Vernier is from the 0 mark on the scale; then note the number of divisions on the Vernier from 0 to a line which exactly coincides with a line on the scale.

Fig. 11—Enlarged Vernier Scale

In Fig. 11, the Vernier has been moved to the right one and four-tenths and one-fortieth inches (1.425 in.), as shown on the scale, and the eleventh line on the Vernier coincides with a line on the scale. Eleven-thousandths of an inch are, therefore, to be added to the reading on the scale, and the total reading is one and four hundred and thirty-six thousandths inches (1.436 in.), which is the distance the jaws have been opened. Because of the extreme accuracy required in most work on which a Vernier is used, many skilled machinists employ an eyeglass when reading the scale. By so doing, the possibility of error is greatly reduced.

HOW TO READ A VERNIER MICROMETER

Readings in ten-thousandths of an inch are obtained *on the Micrometer* by the use of a Vernier, which operates on the same principle as the Vernier on the caliper. In this case, however, ten divisions on the sleeve occupy the distance of nine divisions on the thimble. The difference between the width of one of the ten spaces and one of the nine spaces is one-tenth of a division on the thimble.

Each division on the thimble represents one-thousandth of an inch, and one-tenth of one-thousandth equals one ten-thousandth. To read a ten-thousandth micrometer, first note the thousandths as in the ordinary micrometer. Then observe the line on the sleeve which coincides with a line

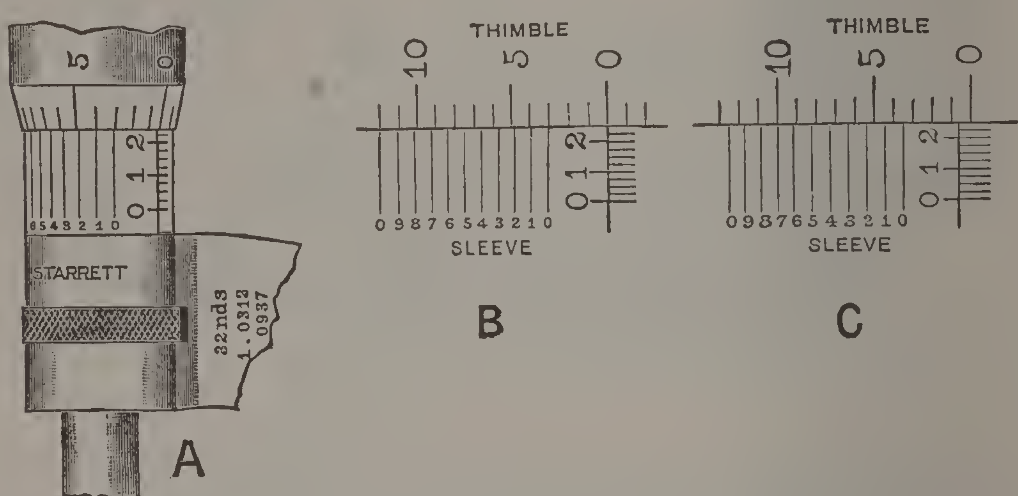


FIG. 12—VERNIER MICROMETER SCALE

on the thimble. In Fig. 12 there are nine vertical divisions visible on the sleeve, and 9 x 25 equals 225, so that the reading of the ordinary micrometer would be .225. Line marked "7" on the sleeve coincides with a line on the thimble and, therefore, we add seven to the reading of the ordinary micrometer. This seven is seven ten-thousandths (.0007), and the reading will be .2257.

OPERATION AND ADJUSTMENT OF MICROMETERS

QUICK MEASUREMENTS.—Some micrometers have a quick-adjustment feature and can be instantly opened or closed to any size within the capacity of the tool. On such tools, pressure of the finger on the end of the plunger allows the spindle to move instantly to approximately the desired size without turning the thimble. When the finger is removed, fine adjustments may be made in the usual way.

MICROMETER AS A GAGE.—Most micrometers have a knurled lock nut, by means of which the spindle can be firmly fixed in position, making the micrometer a solid gage. While this is common practice, the tool should not be used

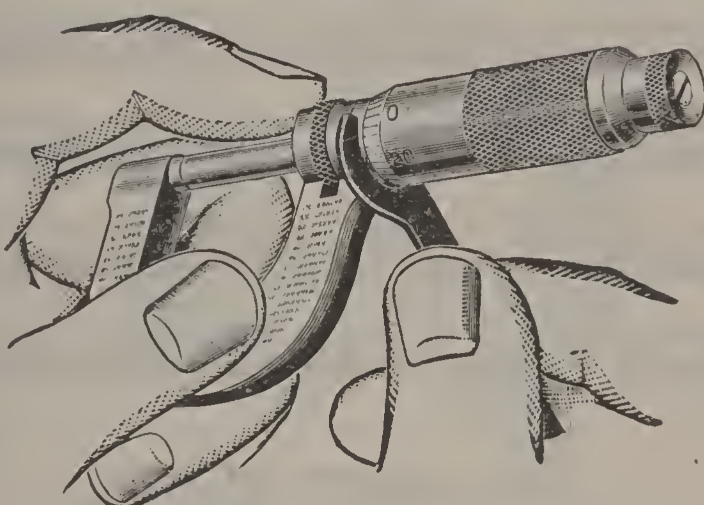


FIG. 13—ADJUSTING FOR WEAR

for this purpose. Instead, gages made expressly for such work and known variously as "go and not go", "snap" and "limit" gages, should be employed. In one type of micrometer, turning the lock nut contracts a split bushing around the spindle, keeping it central and true, while in others, a cam principle is used, the ring and lock nut being set in the frame.

READJUSTMENT FOR WEAR.—When slight wear makes correction necessary, the readjustment is accomplished

by various means and degrees of ease and accuracy, according to the make of micrometer. With the Starrett micrometer the anvil is fixed, not movable, and correction is quickly made by inserting a spanner wrench in the friction sleeve under the thimble and turning until the line on the sleeve coincides with the zero on the thimble when the micrometer is closed or is against a standard plug (See Fig. 13). This adjustment feature does away with all need for the frequent use of a test piece.

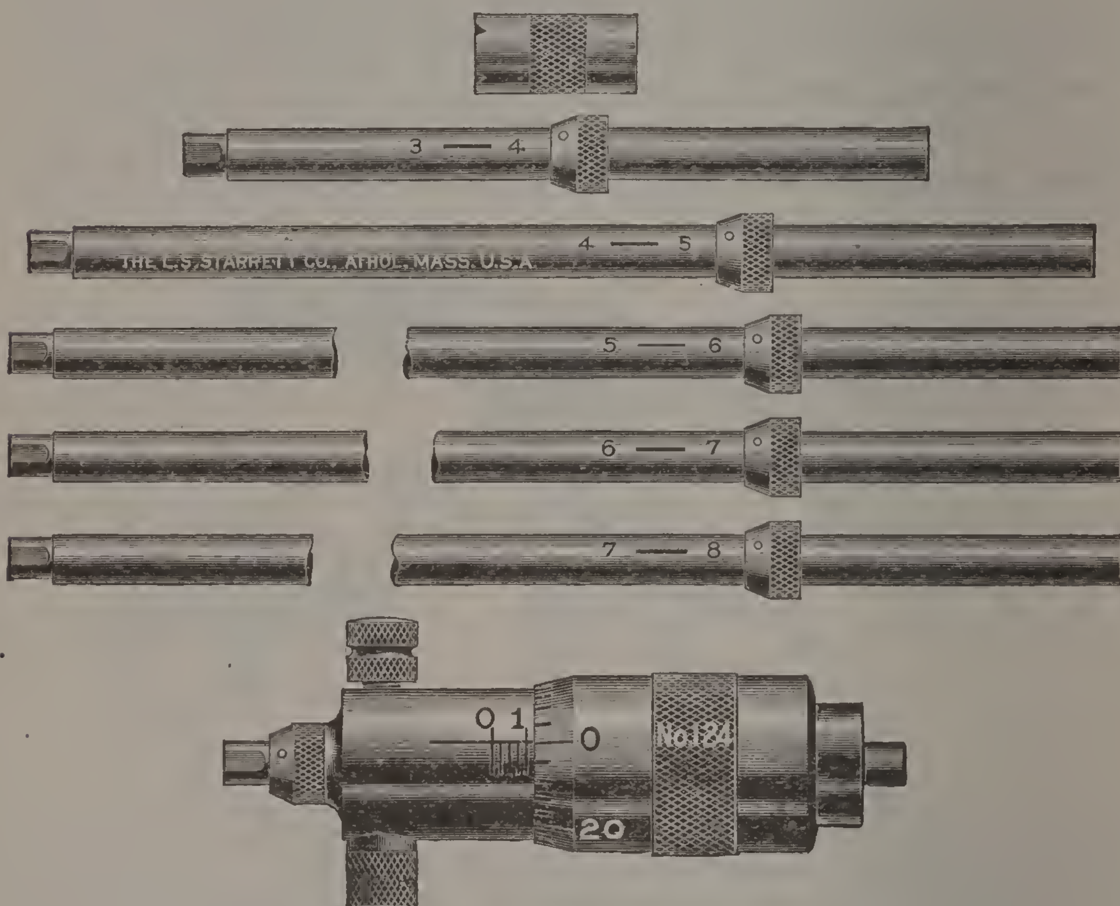


Fig. 14—Inside Micrometer (Starrett)

INSIDE MICROMETERS

In addition to outside measurements with micrometers, automobile work requires the very exact measurements of inside diameters such as cylinder bores, etc. Inside micro-

meters are instruments employing the micrometer principle in such a way that inside diameters can be taken. An inside micrometer may be used to determine whether cylinders have worn oval or have become tapered from top to bottom, though the Starrett Cylinder Gage is far preferable for this particular sort of work.

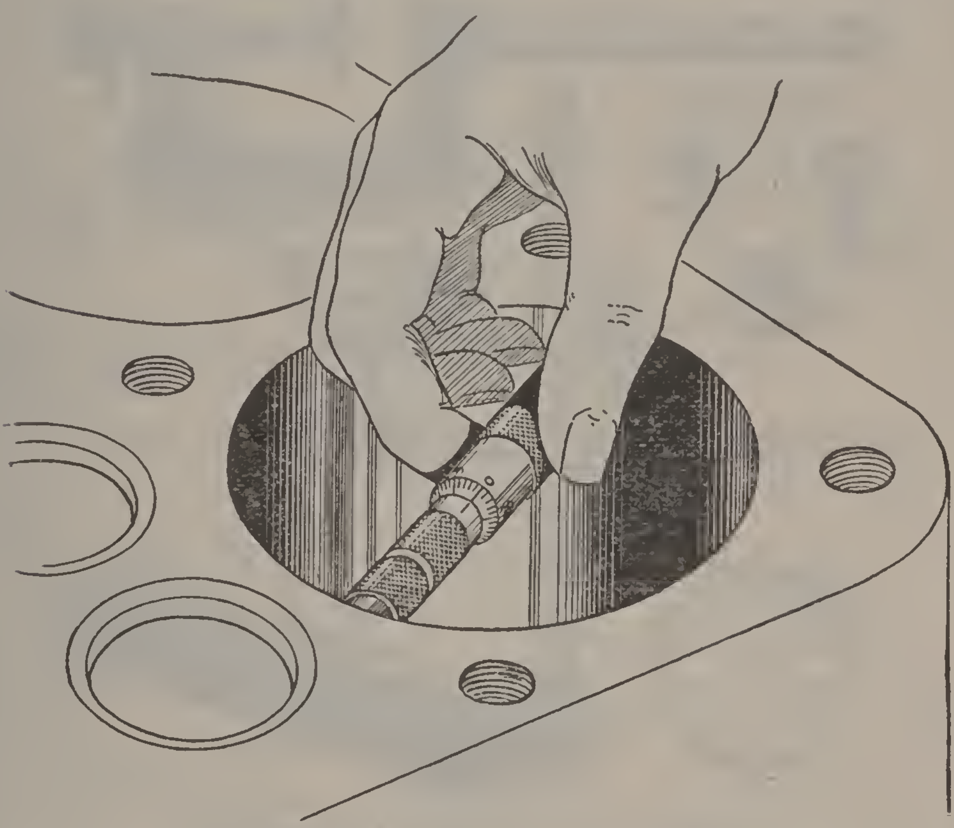


Fig. 15—Measuring a Cylinder Bore with Inside Micrometers

The micrometer screw in the head of the inside micrometer shown in Fig. 14, has half inch movement, and by means of the extension rods measures bores from 2 to 32 inches in diameter by thousandths of an inch. The extension rods are provided with a collar against which the rods are conveniently and accurately set in the micrometer head. In set-

ting, care must be taken to see that the zero mark on the collar coincides with the zero mark on the micrometer head.

Micrometers are made in a wide range of sizes, styles and purposes. Starrett micrometers are made to read in either English or Metric measure, reading in English measure in thousandths or ten-thousandths of an inch and with a range in size—capacity—from 0 in. to 24 in., and in Metric measure in millimeters, the range being from 0 to 150 m.m.

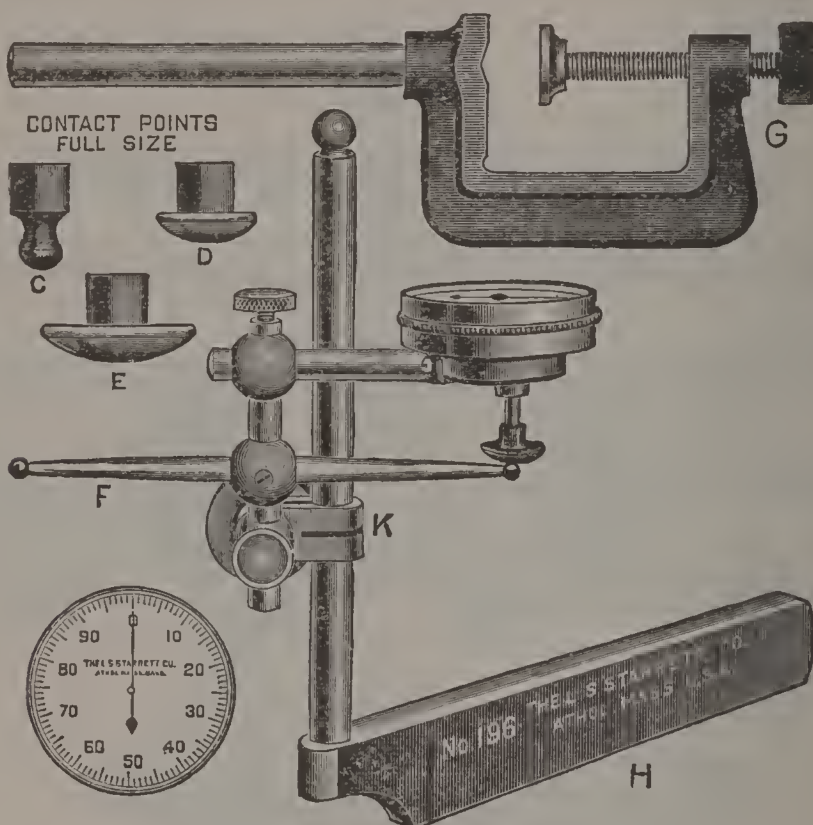


Fig. 16—Dial Gage (Starrett)

DIAL GAGES

Primarily, dial gages are not instruments for making measurements as they do not ordinarily directly indicate distance. They do, however, indicate differences in sizes within their range. In combination with a micrometer, however, they can be used to measure exact distance. The dial gage is a great help in testing shafts for alignment, for testing cylinder

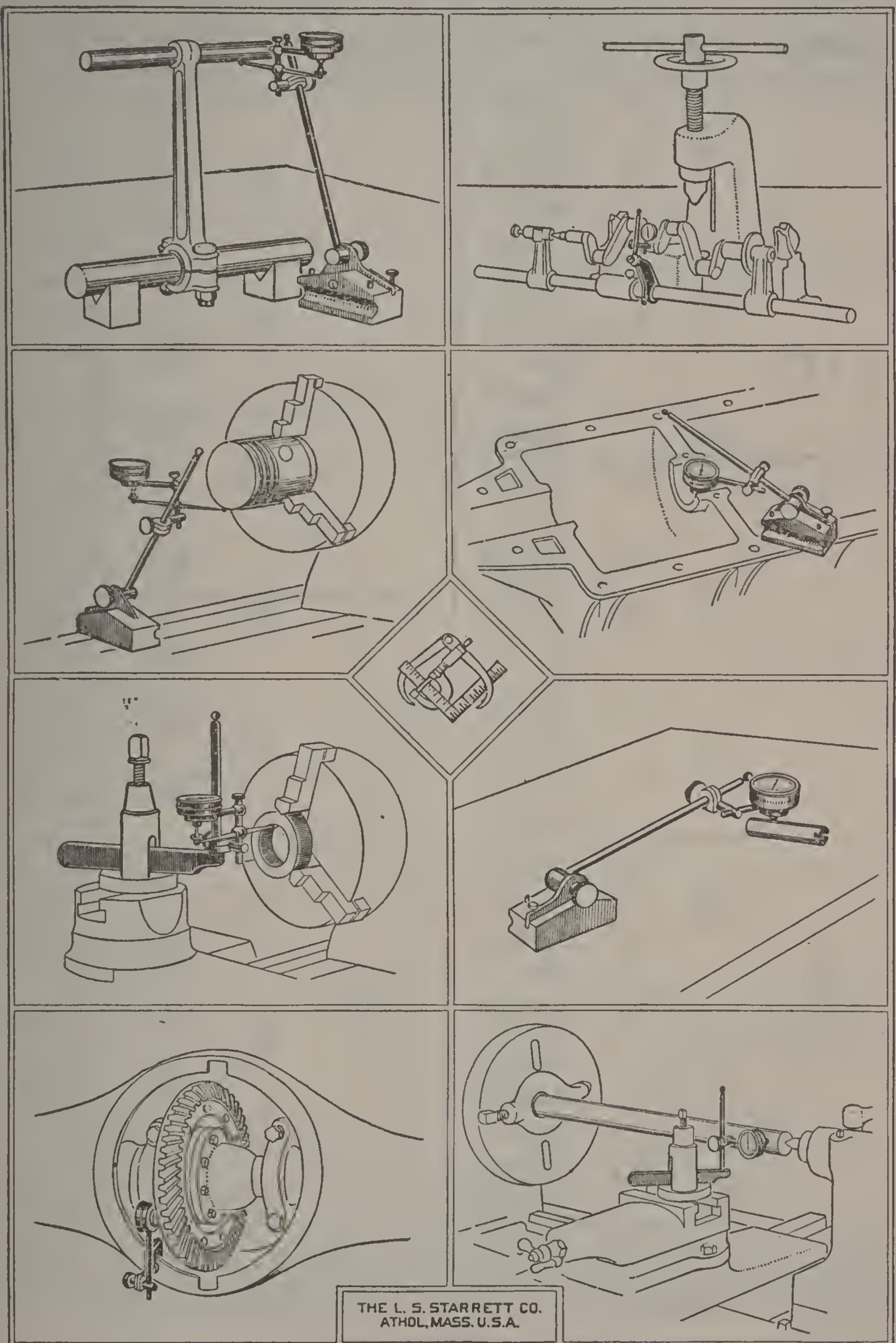


FIG. 17—APPLICATIONS OF DIAL TEST INDICATOR

DECIMAL EQUIVALENTS OF FRACTIONS OF AN INCH*

			1/64	.015625
		1/32		.03125
			3/64	.046875
	1/16			.0625
			5/64	.078125
		3/32		.09375
			7/64	.109375
1/8				.125
			9/64	.140625
		5/32		.15625
			11/64	.171875
	3/16			.1875
			13/64	.203125
		7/32		.21875
			15/64	.234375
1/4				.25
			17/64	.265625
		9/32		.28125
			19/64	.296875
	5/16			.3125
			21/64	.328125
		11/32		.34375
			23/64	.359375
3/8				.375
			25/64	.390625
		13/32		.40625
			27/64	.421875
	7/16			.4375
			29/64	.453125
		15/32		.46875
			31/64	.484375
1/2				.5
			33/64	.515625
		17/32		.53125
			35/64	.546875
	9/16			.5625
			37/64	.578125
		19/32		.59375
			39/64	.609375
5/8				.625
			41/64	.640625
		21/32		.65625
			43/64	.671875
	11/16			.6875
			45/64	.703125
		23/32		.71875
			47/64	.734375
3/4				.75
			49/64	.765625
		25/32		.78125
			51/64	.796875
	13/16			.8125
			53/64	.828125
		27/32		.84375
			55/64	.859375
7/8				.875
			57/64	.890625
		29/32		.90625
			59/64	.921875
	15/16			.9375
			61/64	.953125
		31/32		.96875
			63/64	.984375

*For other decimal equivalents, tables of weights and measures, squares, cubes and square and cube roots, etc., see pp. 165–169, Vol. II, Starrett Books.

WRENCH SIZES FOR BOLTS, NUTS AND CAP SCREWS

Milled Opening in Wrench	U. S. Std. Machine Bolt Nuts	U. S. Std. Machine Bolts	U. S. Std. Bolts and Nuts	U. S. Std. Cap Screws	S.A.E. Bolts Nuts and Cap Screws	Stove Bolt Nuts	Carriage Bolt Nuts	Machine Screw Nuts No.
1/4	—	—	—	—	—	—	—	4
5/16	—	—	1/8	1/8	—	1/8	—	6
11/32	—	—	—	—	—	5/32	—	8
3/8	1/4	—	—	3/16	—	3/16	3/16	10
13/32	—	—	3/16	—	—	—	—	11
7/16	—	1/4	—	1/4	1/4	7/32	1/4	12
15/32	5/16	—	—	—	—	—	—	13
1/2	—	—	1/4	5/16	5/16	1/4	—	14
9/16	3/8	5/16	—	3/8	3/8	5/16	5/16	18
19/32	—	—	5/16	—	—	—	—	20
5/8	—	3/8	—	7/16	7/16	3/8	3/8	22
21/32	7/16	—	—	—	—	—	—	24
3/4	—	7/16	—	1/2	1/2	—	7/16	26
25/32	—	—	7/16	—	—	—	—	30
13/16	—	1/2	—	9/16	—	—	1/2	—
*7/8	—	—	1/2	5/8	9/16	—	—	—
15/16	—	—	—	—	3/8	—	—	—
31/32	—	—	9/16	—	—	—	—	—
1	—	—	—	3/4	—	—	—	—
1 1/16	—	—	5/8	—	3/4	—	—	—
*1 1/8	—	—	—	7/8	—	—	—	—
1 1/4	—	—	3/4	1	7/8	—	—	—
1 3/8	—	—	—	1 1/8	—	—	—	—
1 7/16	—	—	7/8	—	1	—	—	—
1 1/2	—	—	—	1 1/4	—	—	—	—
1 5/8	—	—	1	1 3/8	1 1/8	—	—	—
1 13/16	—	—	1 1/8	—	1 1/4	—	—	—
2	—	—	1 1/4	—	1 3/8	—	—	—
2 3/16	—	—	1 3/8	—	1 1/2	—	—	—
2 3/8	—	—	1 1/2	—	—	—	—	—
2 9/16	—	—	1 5/8	—	—	—	—	—
2 3/4	—	—	1 3/4	—	—	—	—	—
2 15/16	—	—	1 7/8	—	—	—	—	—
3 1/8	—	—	2	—	—	—	—	—
3 1/2	—	—	2 1/4	—	—	—	—	—
3 7/8	—	—	2 1/2	—	—	—	—	—
4 1/4	—	—	2 3/4	—	—	—	—	—
4 5/8	—	—	3	—	—	—	—	—

*Wrench sizes for S. A. E. standard spark plugs.

NOTE: Wrench openings are milled 1/64 in. above the nominal wrench opening given in the first column up to 1¼ in. Above this size the clearance is 1/32 in. or even more in the still larger sizes. Set screws take the same size opening as the diameter of the screw.

bores for roundness and taper and for testing bearing bores. In any of these jobs, the dial gage indicates directly to within .001 in. the alignment or roundness of the article being tested. In the hands of a skilled man, the instrument can be read to within .00025 in. The dial gage is extensively used in manufacturing and is rapidly being found a necessity in service and repair work.

Dial Test Indicators may be used to advantage in connection with straightening crankshafts, locating wrist pin holes, determining amount of shim to insert or remove, determining taper, checking play in bearings, reboring work, lining up Ford Magneto Coil Assembly, etc.

FILING*

The file, one of the most common tools in the shop, is a cutting tool with a great number of fine, sharp edges or teeth which do the cutting when the file is moved forward across the work. Files are made in a great variety of shapes, sizes and tooth grades to meet various requirements.

The more common shapes are hand, flat, mill, round, half round, square, pillar, three square, taper and slim taper. The lengths range from 3 in. to 20 in. or more. The following would be a typical selection for an automobile shop:

Hand bastard, 8, 10 and 12 in.
Flat bastard, 8, 10 and 12 in.
Flat smooth, 10 and 12 in.
Half round second cut, 6, 8 and 10 in.
Mill second cut, 8 and 10 in.
Square second cut, 6, 8 and 10 in.
Taper second cut, 3, 4, 6 and 7 in.
Round second cut, 3, 4, 6 and 8 in.
Hand rough, 10 and 12 in.

The size of the teeth or cut starts with the coarsest to the finest, rough, bastard, second cut, smooth and dead smooth. The relative sizes of the teeth are shown in the illustration.

*See also pp. 40–42, Vol. I, Starrett Books.

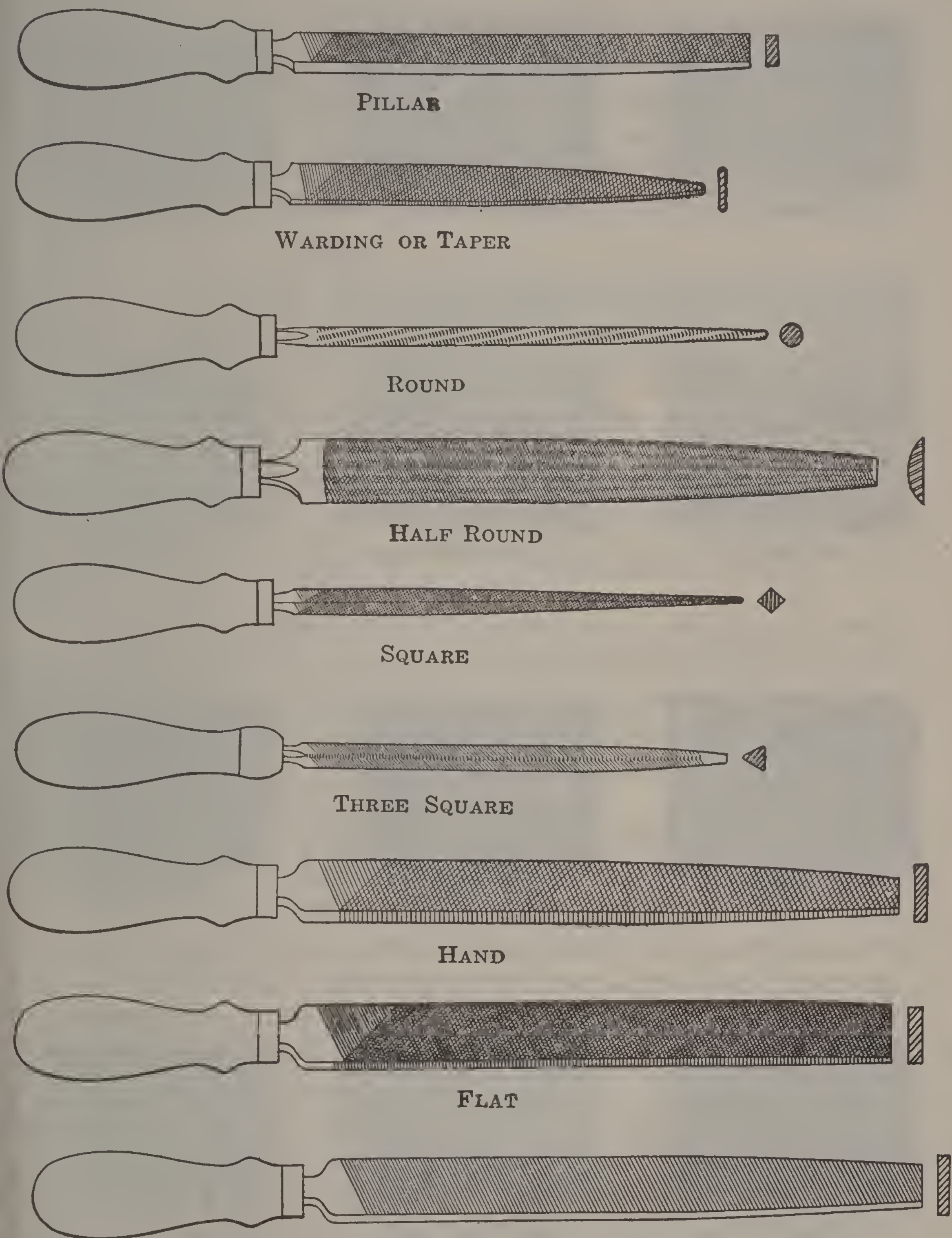


Fig. 18—File Shapes

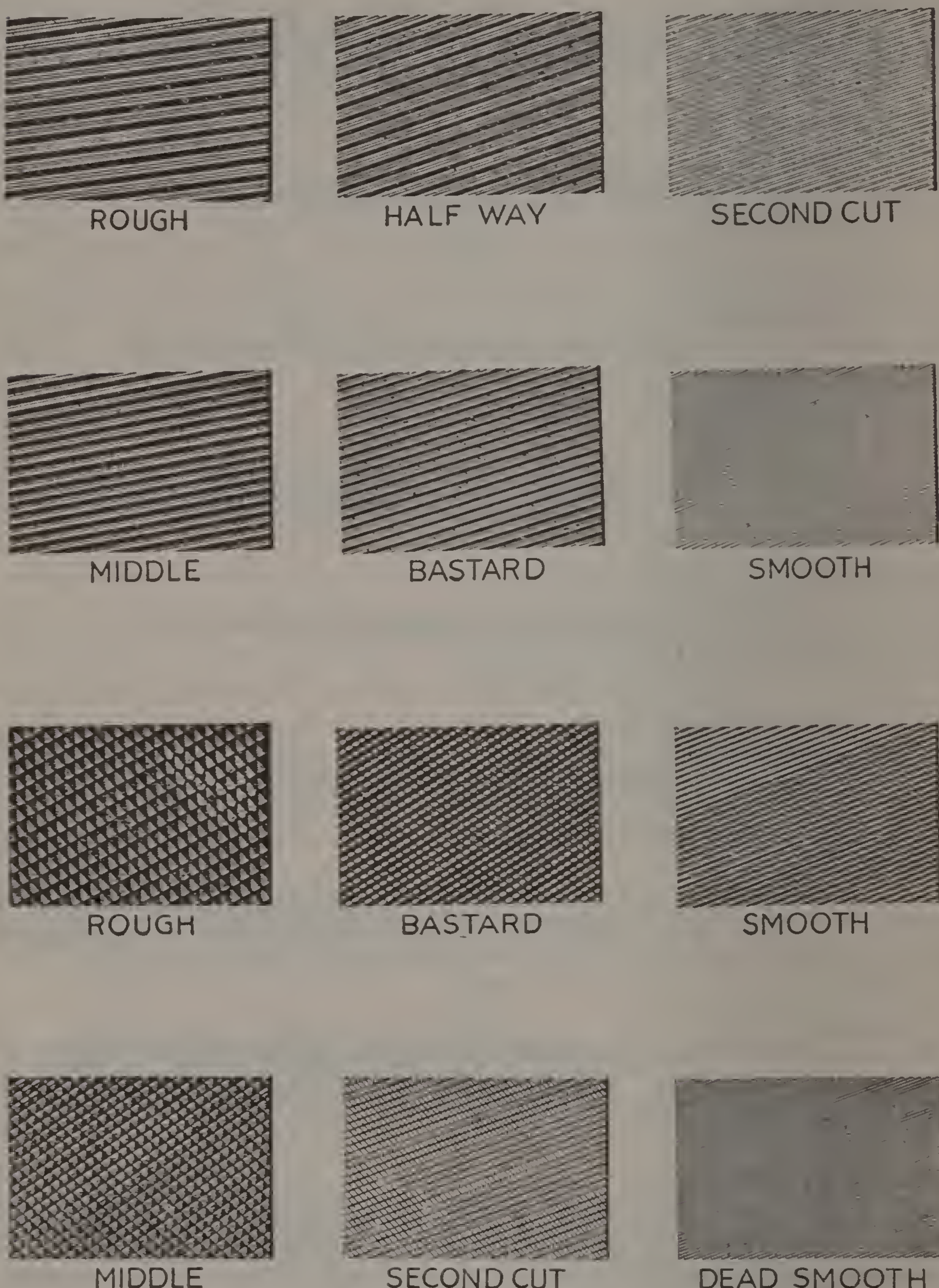


Fig. 19—File Cuts

The teeth vary in the same cut with the size of the file, a bastard cut being larger in a 10 in. file than in a 6 in. file.

Most of the cuts are to be had in either single or double cut. A single cut file has the teeth cut across in one direction only. Double cut files have the teeth cut in two directions across the file, the cuts being at a considerable angle

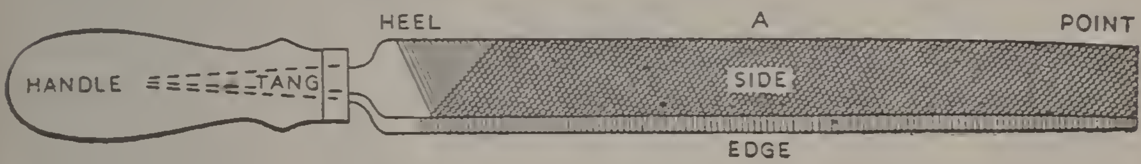


Fig. 20—Parts of a File

to each other. Double cut files are to be preferred for all general shop work as they do better work and last longer. The single cut files are useful for finishing up fine work and for draw filing.

The length of a file is from the heel to the point, exclusive of the tang. The point is the end farthest from the tang and the heel is the juncture of the tang and the file proper. Some files have one or more edges or sides without teeth. These are known as "safe" sides or edges and are useful in fitting step joint piston rings, etc.

Files are made of high grade carbon steel, hardened and tempered. In the hardening process it is quite common for the file to attain a very slight curvature which can be used to advantage when the mechanic wants to file perfectly flat or slightly hollow. Sighting along the file will tell how any particular file is curved and which side to use.

Much better work can be done with a file if it is properly handled. Patent iron or steel handles with quick acting fastenings are fast finding favor. Wooden handles should be in proportion to the size of the file. The handle should be put on straight and the tang should go into the handle almost up to the heel. To avoid splitting the handle, heat the tang of an old file of the same size to a red heat and drive the handle on, the tang burning its way in. When

nearly to the desired point, the handle is pulled off, and the new file, first having been dipped in water, is driven on tight. A file handled this way will stay handled.

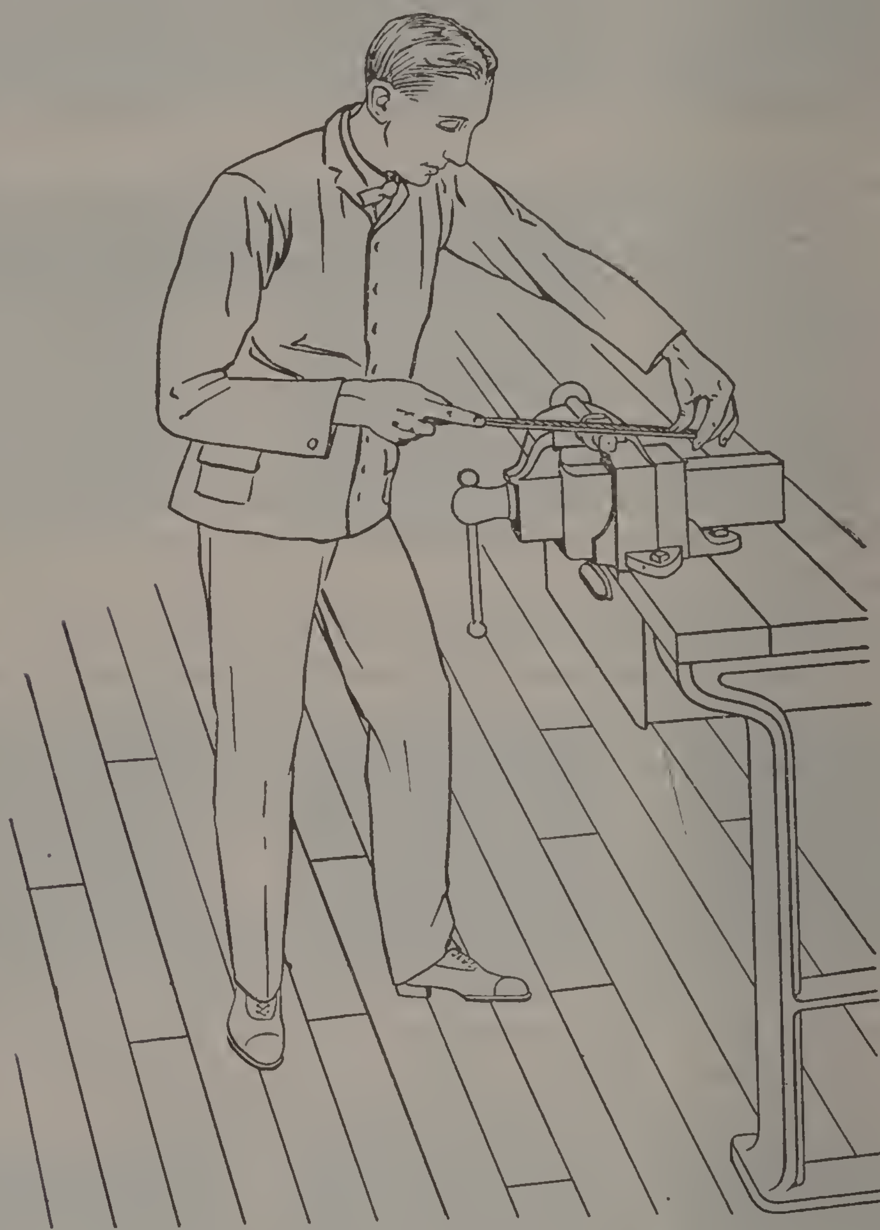


FIG. 21—FILING

The proper height for filing depends on the height of the mechanic. When he stands straight and in a filing position, if the right forearm and file are in a straight line the position is about perfect. For the average man this height is about

42 in. If the mechanic is short he should use a wooden platform to bring his elbow to the height of the work.

Files will last longer and give better service if they are properly taken care of. Being highly tempered, the teeth are quite brittle and, therefore, files should never be thrown around on a concrete floor nor thrown one on top of the other. It is preferable that there be racks to hang them up when not in use. This will also keep them out of oil and grease which are bad for files and prevent their cutting, although oily files can be cleaned with gasoline. It is better to use a file exclusively on one metal. Files for brass might have red handles, for iron and steel black handles and for babbitt and lead, varnished or natural handles.

When files become clogged with bits of metal, they will scratch and they should be cleaned with a file card which is a sort of wire brush with tiny bent wires. A file which screeches can be silenced by rubbing chalk in it. In finish filing the use of chalk or oil will eliminate "bugs". Use old, worn out files for lead and babbitt as these metals will clog up a sharp file. Worn out files can be recut, but unless they are large and expensive the cost of recutting is hardly worth while. The teeth of an old file can be sharpened slightly by first removing all oil or grease from the file with gasoline and then soaking it for four to eight hours in dilute sulphuric acid. The teeth, however, while somewhat sharper will not be exactly even.

HOW TO USE HACK SAWS*

The hack saw is one of the essential tools of the automobile shop. It is used to cut a great variety of metals and other substances and in many shapes. Here are some of the jobs ordinarily encountered: Sawing out rusted bolts and studs; cutting old bronze bushings, new bolts, drill rod, etc.; sawing off brass and copper gasoline and oil pipes; cutting screwdriver slots in cap screw heads; sawing through chassis

*See also "Hacksaws and Their Use," The L. S. Starrett Co., publishers. Also pp. 43–46, Vol. I, Starrett Books, and pp. 103–106, Vol. II, Starrett Books.

frames for extensions and through old solid rubber tires and rims for removal; undercutting commutator mica; sawing fiber sheets and blocks for electrical work; removing and replacing piston rings; sawing lead battery connectors and posts; sawing slots in exhaust pipes for cut-outs and car heaters; sawing cast iron pistons for cylinder laps.

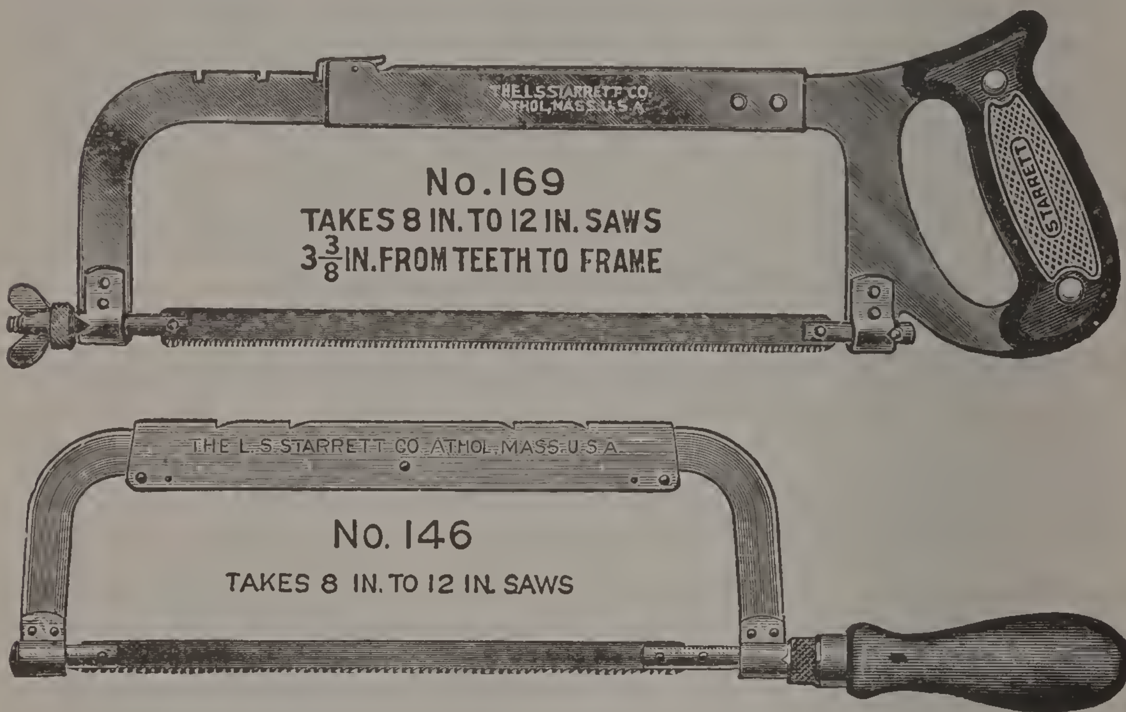


Fig. 22—Hacksaw Frames (Starrett)

Practically all of this work is done with hacksaws used in a hand frame with blades 8, 10 or 12 in. long. In the larger shops where a considerable quantity of sawing is done it will pay to use a power hacksaw. This is especially true where the shop does assembling, partial manufacturing, body building, etc.

The life of a hacksaw practically ends when a tooth breaks. After the first tooth goes, an additional strain is placed on the tooth in back of the broken one and it is only a matter of a short time until a number of teeth are missing and the saw ruined as a cutting tool. While hacksaw blades are comparatively cheap, much money can be saved by using the saws to best advantage and prolonging their life as much

as possible. Using the proper blade in the proper manner will increase the life of the saw several hundred per cent and save an incalculable amount of time and effort.*

Hacksaw frames should be stiff and rigid so that there is no frame weave or movement on either the forward or back strokes. If of the extension type, there should be no bowing of the frame's back when tension is placed on the blade. The handle should fit the user's hand comfortably and—unless of the regulation "saw-handle" or "easy" type—should be straight with the rest of the frame. The pins which hold the blade in place should be in tight and should have enough rake to keep the saw from slipping off. Hacksaw blades should always be so placed in the frame that the rake of the teeth is forward—that is, away from the operator.

Before using, the saw should be strained or tightened so that it gives a sort of musical twang when plucked like a fiddle string. A loose saw will break easily. A saw too

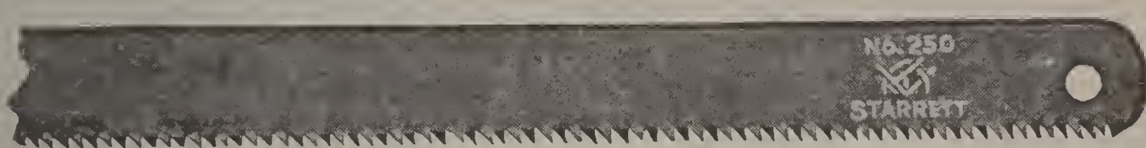


FIG. 23—HACKSAW BLADES (STARRETT)

tight is liable to break the blade. Looseness in the blade often causes a slight twist in the cut.

Too much care cannot be exercised in the choice of the saw used. Different materials and different shapes require blades of varying thickness or gage, as well as a varying number of teeth to the inch. To insure the economical, efficient use of hacksaws consult the Starrett Hacksaw Chart for Automobile Shops (Page 33).

When using hacksaws be sure to put sufficient pressure on the saw to make the teeth cut. If the saw lazes across the

*See page 33.

NOTE: For information as to the proper blades to use in ordinary machine shop work see the regular or Standard Starrett Hacksaw Chart, Vol. II, Starrett Books.

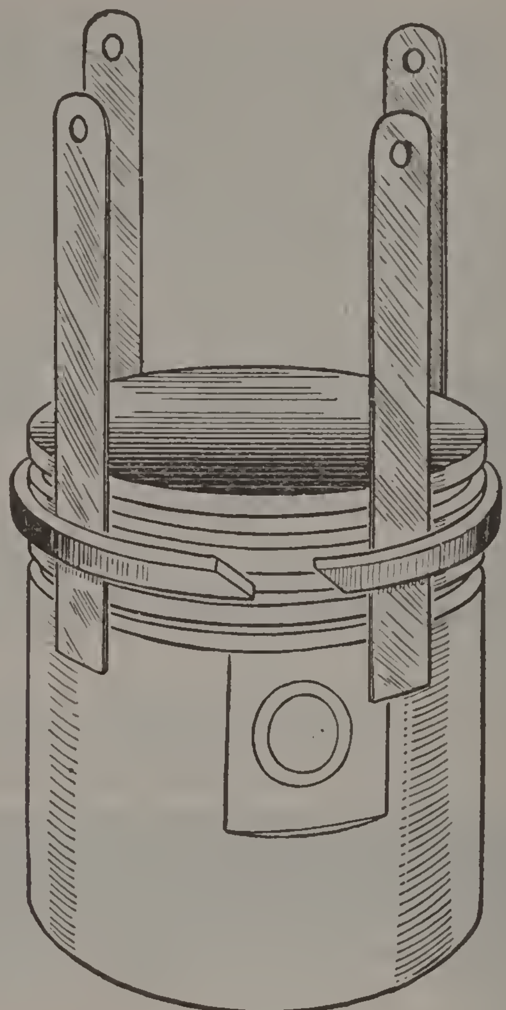


Fig. 25—Hacksaw Blades Used to Remove Piston Rings

work without cutting, the teeth are dulled rapidly, shortening the life of the blade. Where practical, release the pressure on the back stroke. Make the stroke as near the full length of the blade as possible. Do not attempt to cut through hardened or tempered steel and go cautiously on welded iron or steel work. The rapid chilling of the weld often makes the metal nearly glass hard. When cutting a weld or tempered steel is unavoidable, the metal should—whenever possible—first be heated to a cherry red and cooled slowly, the work being handled as in annealing and the metal not allowed to cool in air.

STARRETT HACKSAW RECOMMENDATIONS FOR AUTOMOBILE WORK

1. Sawing out rusted bolts and studs, diameters from ¼ to ¾ in., cold rolled steel, nickel steel and molybdenum steel.	No. 250 and No. 250-B
2. Sawing out worn bronze bushings.	No. 250-B
3. Sawing new bolts, studs, drill rod, etc.	No. 250 and No. 250-B
4. Sawing brass and copper gasoline and oil pipes, diameters inside ¼ and 5/16. Thickness about 1/16 to 3/32. Annealed.	No. 252 and (No. 258 for under 1/16″ thick)
5. Sawing screw driver slots in cap screw heads, cold rolled and nickel steel.	No. 115 and No. 249-A-B-C-D
6. Sawing through chassis frames for extensions, patching, etc.*	No. 250 and No. 250-B
7. Sawing through old solid rubber tires and steel rims for removing when hydraulic press is not available	No. 115 and No. 115-B No. 262 and No. 259
8. Undercutting commutator mica. . . .	No. 103-No. 112
9. Sawing fiber sheets and blocks for electrical work..	No. 103-No. 112. All hard saws 32 to 18 point.
10. Sawing lead battery connectors and posts. Metal is about 98 per cent lead with a little antimony for hardness.	No. 250-B
11. Sawing slots in exhaust pipes for installing cut-outs and car heaters. The pipes are from 1¼ to 3 in. outside diameter and from 1/16 to ⅛ in. thick. The slots are usually sawed V-shape about halfway through the pipe. In some cases the cut is square with the pipe, a diagonal being sawed and the flaps bent out and cut off.	No. 258 for 1/16″ thick No. 252 for 3/32″ thick No. 250 for ⅛″ thick
12. Sawing through cast iron pistons to make cylinder laps.	No. 250-No. 115-No. 262

*NOTE: No. 6 is same as No. 1 and applies if the frames are of *molybdenum* steel.

Old hacksaw blades may be saved and the teeth ground off so that they can be used behind piston rings for removing and installing.

REAMERS AND REAMING*

There are two general classes or divisions of reamers; those employed in only the rougher kinds of work, and those used in the production and repair of high grade machinery. Hand reamers, such as are used in automobile service station and repair shop work, belong in the second class and are really high grade precision tools.

Fig. 26—Reamer

There are three types of hand reamers which have come to be generally accepted by mechanics and machinists as standard. The first is the straight, solid, fluted reamer with regularly spaced flutes (See Fig. 26).

This is the oldest and is still the most common type of reamer in general machine shop practice, although in automobile repair work reamers of the expanding type have largely displaced it. The solid reamer is a rugged tool, made from a single piece of steel in which are cut longitudinal grooves or flutes providing the necessary high sections or "lands". These "lands", or cutting edges, are hardened, tempered and carefully ground to very close limits. For accurate work this type of reamer cannot be excelled, but its rapid loss of size through wear and its lack of flexibility places it at a disadvantage and accounts for the popularity of expansion or adjustable type reamers. The solid reamer, while originally made with straight, regularly spaced flutes, is now often found with irregularly spaced flutes to avoid

*See page 98, Vol. I, Starrett Books, and pp. 99-101, and 114-121, Vol. II, Starrett Books.

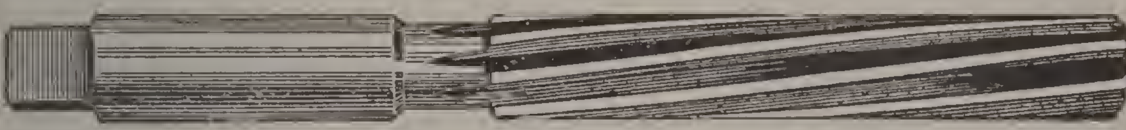


Fig. 27

Fig. 28

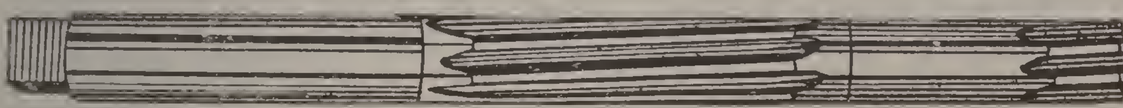


Fig. 29

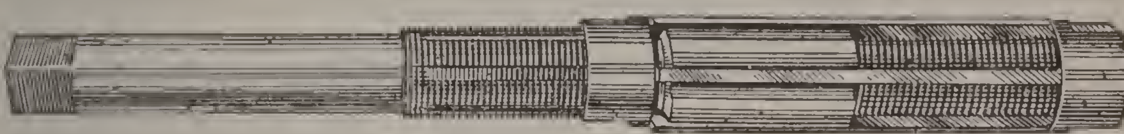


Fig. 30

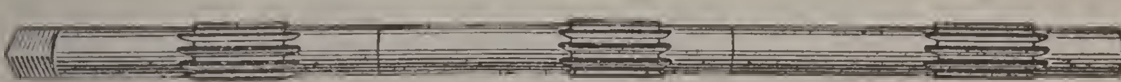


Fig. 31

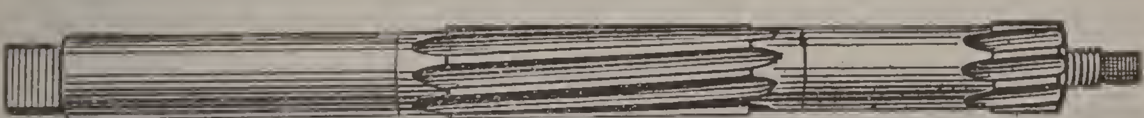


Fig. 32

Fig. 33

Fig. 34

"chatter", which will cause a roughness in the finish of the work, and is also made in spiral form as shown in Fig. 27. This tool is often used to advantage when reaming bearings having an oil groove or other open space.

The expansion reamer (See Fig. 28), as its name indicates, is a semi-solid tool so made that it may be expanded a few thousandths of an inch by means of a taper screw plug. This permits compensation for a limited amount of regrinding. The tool, however, does not give the extreme accuracy of the solid reamer.

The adjustable reamer, often called the Critchley Type reamer (Fig. 30), is entirely different in principle and construction from the types already described. In this type the blades are separate from the body of the tool and are mounted in staggered grooves taper milled in the shank of the tool, which is threaded to take adjusting nuts that serve to move the blades up or down the grooves, increasing or decreasing the diameter of the reamer. The great flexibility of the tool, coupled with the fact that the blades may be repeatedly reground or replaced when necessary, has won for it great popularity with the motor machinist and auto repair man. Its disadvantages are that its construction is such that the tool is easily injured. It must be kept clean and requires great care in its use if anything approaching accurate work is to be done.

All these types of reamers—the solid, expansion and adjustable, or Critchley—are also manufactured with pilots to assist in guiding the reamer in various kinds of work, especially where it is necessary to ream two or more bushings or bearings in perfect alignment. Among reamers equipped with pilots are the solid reamer with starting and aligning pilots, the crankshaft bearing reamer (Fig. 31), the spiral reamer with starting and aligning pilots (Fig. 32), the spindle bolt bushing reamer with short flutes and rear pilot (Fig. 33) and expansion reamers with front and rear pilots (Fig. 34).

In general machine shop practice a reamer is ordinarily used to finish a hole true to size after first boring it with a drill

or with a boring bar on a lathe or some other machine tool. Often an arbor, or mandrel, will be forced into this hole and other surfaces finished on the lathe, using the centers which are in the arbor. This is a very common job to lathe workers and is pretty well understood by machinists. In automobile work, however, it is seldom that this sort of work is performed excepting in cases of emergency where a special part has to be made to order.

Practically all parts supplied by manufacturers through distributors, dealers and agents fit into the car or the unit without further machine work or fitting, with the exception of bearings and bushings. These latter have to be fitted

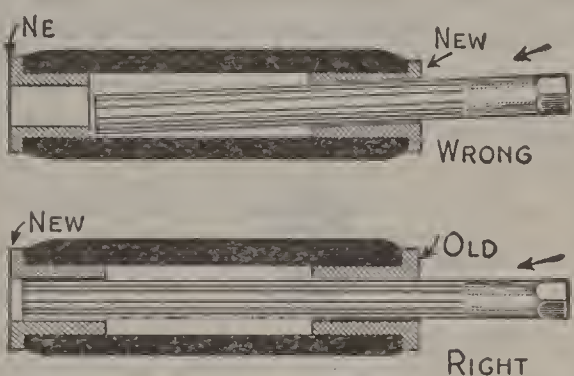


FIG. 35

by the mechanic by scraping, reaming or lapping. The process used depends on the character of the bearings.

In the case of bronze bushings, the fitting is done with reamers which finish cut the holes in the bushings to an exact fit for the pin or shaft, etc., that is to work in the bushing. For that reason, it is customary to replace the pin as well as the bushing, because they both wear. The rebushed part will then have parts that fit in every respect as they did when the car was new.

Where a bronze bushing is by itself and does not have to line up with anything else, it is simply driven in place with an arbor press and the hole carefully reamed. Spring eye bushings would be examples of this kind.

Where, however, the bronze bushings are in pairs and have

to line up with each other, different methods have to be used. Bronze bushings are in pairs in the steering spindles, in the pistons, in many cases as bearings for the wrist pins, and sometimes in the gear-case. Camshaft and crankshaft bushings and bearings also call for alignment in reaming.

In reaming bearings opposite each other, or paired, a process is used which is called align reaming. The type of reamer to use on this work has already been described and is called an aligning or "align" reamer. The illustrations show how an align reamer works and what sort of hole might result if an ordinary reamer were used (See Fig. 35).

Reamers are also used in places other than bearings. When valve stems wear loose a reamer a few thousandths of an inch oversize is run through the stem hole and a new valve with an oversize stem inserted. Where the stem guide is removable the old one is pressed out, a new one pressed in

Fig. 36—Valve Reseater

and a reamer run through. A valve seat reamer with teeth at an angle of 45 degrees—or whatever may be the correct valve angle for the particular make of motor, trues up the valve seat preparatory to grinding, thus saving considerable time and insuring a better seat. This tool is sometimes called a valve reseater, but it is really a special reamer. The better types have two cutting faces, one of which should be used for the roughing cut, removing the glaze from the valve seat, the other being reserved for the finish cut (Fig. 36).

Taper pin reamers are used to ream tapered holes to fit standard taper pins. These are often used in such places as starting crank pins, pins to hold magneto and generator couplings to lay shafts, etc. Standard taper pins and reamers have a taper of ¼ in. per ft. The numbers run from 0000, which is the smallest, to 14, which is the largest. The sizes

of taper pin reamers usually met with in automobile work range from 0 to 5 or 6. A table of standard taper pin and taper pin reamer sizes is given on page 44.

USING REAMERS

All stock reamers of all kinds cut right-handed. That is, they turn the same direction as a drill or as a screw when it is screwed in. The cutting edge of a reamer is so shaped that it has some bearing surface and some clearance. For that reason, a reamer should never be turned backward in the hole. To do so will impose more wear on it than it would receive in reaming a score or more holes.

Reamers are made from good quality tool steel tempered fairly hard. They, therefore, must be handled carefully, not dropped or thrown and never laid or dropped against other tools or reamers. To do so is to run a chance of nicking the sharp edges. Expansion reamers are often broken by trying to force them beyond their normal range of expansion and by expanding them rapidly. Adjustable or Critchley Type reamers must be kept clean in order to work properly or even satisfactorily. Their design is such that dirt, chips, etc., often work underneath the cutter blades, forcing the reamer out of true, causing chattering, digging in, etc. Broken blades, twisted shanks and poor, inaccurate reaming are often the result of dirt on the tool. Critchley Type reamers should be washed with gasoline after use and should be taken apart and properly cleaned. Care must be taken to see that each blade is replaced in the slot from which it was removed.

Reamers should not be expected to take out too much metal at a single cut. For sizes from ¼ to 1 in. the usual allowance is about .003 to .005 in. For sizes larger than 1 in., the allowance is generally a little more. On any job calling for the removal of an excessive amount of metal a drill or boring bar should be used to take out most of the metal before attempting to ream the hole.

Reamers are usually made to feed themselves into the work at the proper speed. Feeding them too fast or too slow will cause chattering and an uneven hole. Spiral fluted reamers feed themselves and straight fluted reamers will feed themselves if used vertically. Otherwise a slight pressure should be used.

The work, when reaming, should be held firmly and in such position as is most convenient for the operator. The use of a bench vise, where size permits, is recommended or in cases where the piece is too large to be handled in this manner, the work should be clamped to the face plate of a drill press with a lathe center in the spindle to take the reamer, which should, however, invariably be turned by hand. A hand reamer should never be operated by power.

Use a tap wrench on a reamer where possible in preference to a monkey wrench, solid wrench or pipe wrench. This will prolong the life of both reamers and wrenches as well as produce better work by distributing the turning force on both sides of the reamer so that there is no side pressure.

Reamers must be started and operated with considerable care in babbitt as the metal is very soft and even a moderate pressure sideways on the reamer will cause it to cut out of line.

When a reamer gets dull it can be sharpened by grinding, but the grinding of reamers should not be attempted except by those who are skilled in the art and have the proper special equipment. The reamer manufacturer or his agent will be glad to perform this kind of work and guarantee a true job.*

REAMER SHARPENING

As has already been stated, reamer sharpening is a factory operation and one which calls for a considerable amount of special equipment. It is, however, practical for the average machinist to restore the edge of a slightly dulled reamer by using a thin hone or abrasive wheel in the flutes,

*For a table of reamer clearances for grinding reamers see Starrett Books, Vol. II, pages 100, 101.

but anything beyond this should be done as closely as possible along the methods followed by the maker of the reamer. In the case of Critchley Type, or adjustable reamers, some manufacturers are now making them with interchangeable blades. This makes it possible to entirely avoid both the use of dull reamers and the uncertain results of attempting to grind them in the shop, since all that is necessary is to buy a new set of blades—exactly as though the tool was a safety razor—drop them into position in the slots in the shank of the tool and get on with the work. All makes of reamers of the Critchley Type, however, do not possess this feature and, of course, new blades cannot be put into either the solid or expansion types of reamer. For that reason, in view of the delay that factory repairs usually entail, it may not be out of place to explain briefly factory methods of

FIG. 38—CROSS SECTION OF SOLID REAMER AFTER ROTARY GRINDING

reamer sharpening for such repair shops, service stations, etc., as have the requisite mechanical equipment.

In factory practice, reamer grinding is divided into two operations; the rotary grinding and the "backing off" or relieving. The rotary grinding is a wet process and differs in no way from other operations of its kind except that it calls for extreme care and accuracy as it is at this stage that the size of the reamer is determined. A cross section of a solid, fluted reamer after grinding is shown in Fig. 38. Note that at this stage the reamer has no cutting quality whatsoever. It is not until the individual flutes are backed off that the reamer becomes a cutting tool.

Backing off, or relieving, determines the efficiency of the tool and is the phase of regrinding in which most machinists fail. Relieving is done by placing the reamer on centers in a tool grinder and using—ordinarily—a cupped wheel. The angle of relief is all-important and is determined by the height of the tooth rest with reference to the center of the reamer. One blade is ground at a time and is held in position by the tooth rest.

The proper angle of relief ranges between 5° and 10°. In automobile repair work 7° will probably be found most satisfactory.

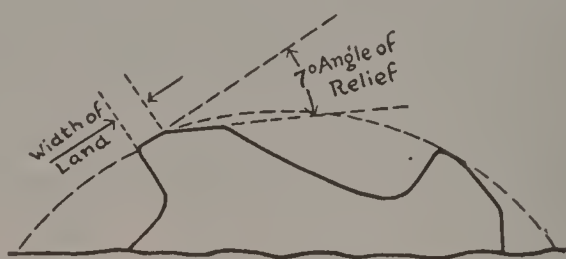


Fig. 39—Section of Solid Reamer after Grinding Clearance on Blade

When relieving, a narrow section should be left between the lip of the blade and the point at which the backing off commences. This is known as the "land" and its width varies from .005 to .025 in. according to the work on which the tool is to be used. Here again a mean will be found most satisfactory for automobile repair work and a "land" of about .010 is recommended. A cross section of a solid reamer backed off in accordance with these recommendations is shown in Fig. 39.

Figure 40 shows the relative positions of the reamer and grinding wheel centers when grinding off the cutting edge. Note that the wheel center is below the reamer center for grinding on the cutting edge to secure greater keenness. Generally speaking, it is safer to run the grinding wheel off the edge as shown in A, but a keener cutting edge may be secured by following the practice indicated in B, although there is danger of the wheel action drawing the tool away from the tooth rest.

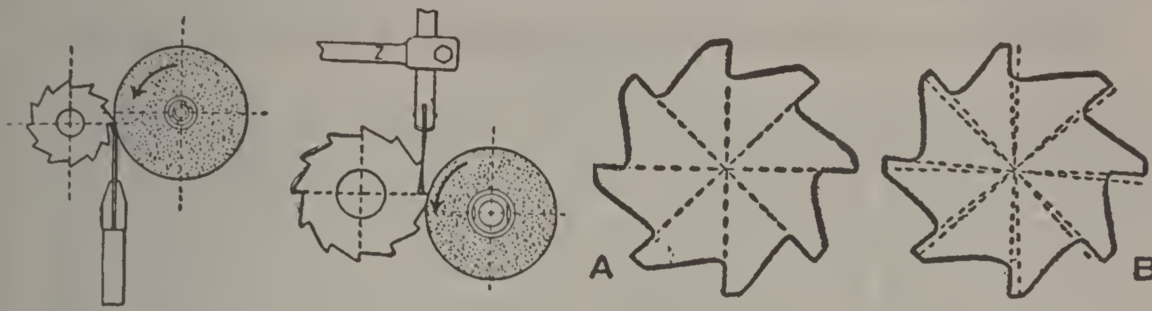


FIG. 40—RELATIVE POSITIONS OF REAMER AND GRINDING WHEEL CENTERS

Table of Drills for Reaming

The drill size is 1/64 in. less than the reamer size in each case, this being the proper amount of stock to leave for reaming under average conditions, providing the drill is correctly ground.

Reamer Size	Drill Size	Reamer Size	Drill Size	Reamer Size	Drill Size
1/4	15/64	9/16	35/64	13/16	51/64
5/16	19/64	5/8	39/64	7/8	55/64
3/8	23/64	11/16	43/64	15/16	59/64
7/16	27/64	3/4	47/64	1	63/64
1/2	31/64				

STANDARD TAPER PINS AND TAPER PIN REAMERS

Taper Pins				Reamers		
No.	Diam. at Large End	Approximate Fraction	Diam. at Small End	Diam. at Large End	Flute Length	Drill Size No.
0000	.109	7/64	.088	.114	1¼	42
000	.125	1/8	.101	.130	1⅜	37
00	.141	9/64	.114	.145	1½	32
0	.156	5/32	.127	.161	1⅝	29
1	.172	11/64	.146	.182	1¾	27
2	.193	3/16	.162	.204	2	21
3	.219	7/32	.183	.230	2¼	15
4	.250	1/4	.208	.251	2½	4
5	.289	19/64	.240	.303	3	1/4
6	.341	11/32	.279	.355	3⅝	9/32
7	.409	13/32	.331	.425	4½	11/32
8	.492	1/2	.398	.507	5¼	13/32
9	.591	19/32	.482	.610	6⅛	31/64
10	.706	45/64	.581	.727	7	19/32
11	.857	55/64	.706	.878	8¼	23/32
12	1.013	1 1/64	.842	1.050	10	55/64

For tables of dimensions of Morse, Jarno, B. & S., Whitney, etc., standard sockets and shanks see pages 114 to 121 of Vol. II, Starrett Books.

TAPER REAMERS FOR STANDARD TAPER SOCKETS

No. of Taper	Total Lgth.	Diameter		Taper, Inch per Foot	Lgth. of Flute	Groove		Shank Diameter	Square	
		Small End	Large End			Depth	B't'm. Width		Side	Lgth.
MORSE										
0	3 3/4	0.250	3.371	0.625	2 1/4	1/40	1/32	11/32	1/4	5/16
1	5 1/2	0.367	0.367	0.600	3	1/40	3/64	15/32	23/64	1/2
2	7	0.569	0.517	0.602	3 1/2	1/32	1/16	21/32	1/2	5/8
3	8	0.775	0.745	0.602	4 1/4	1/32	5/64	7/8	21/32	13/16
4	9	1.017	0.988	0.623	5 1/4	3/64	3/32	1 1/8	27/32	1
5	10	1.471	1.289	0.630	6 1/4	3/64	7/64	1 1/2	1 1/2	1 1/8
6	12	2.112	1.799	0.626	8 1/2	1/16	1/8	2	1 1/8	1 5/16
7	16	2.746	2.555	0.625	12	1/16	9/64	2 5/8	1 31/32	1 1/2
JARNO										
1	1 3/4	0.100	0.1438	0.600	7/8	1/40	1/32	1/8	3/32	3/16
2	2 5/8	0.200	0.2688	0.600	1 3/8	1/40	1/32	15/64	11/64	9/32
3	3 1/2	0.300	0.4000	0.600	2	1/40	1/32	3/8	9/32	3/8
4	4 3/8	0.400	0.5312	0.600	2 5/8	1/40	3/64	15/32	23/64	1/2
5	5 1/8	0.500	0.6594	0.600	3 3/16	1/40	3/64	19/32	29/64	5/8
6	5 7/8	0.600	0.7875	0.600	3 3/4	1/32	1/16	11/16	33/64	5/8
7	6 5/8	0.700	0.9156	0.600	4 5/16	1/32	1/16	13/16	39/64	3/4
8	7 3/8	0.800	1.0438	0.600	4 7/8	1/32	5/64	15/16	45/64	7/8
9	8 1/8	0.900	1.1688	0.600	5 3/8	1/32	5/64	1 1/16	51/64	7/8
10	8 7/8	1.000	1.2969	0.600	5 15/16	3/64	3/32	1 1/8	27/64	1
11	9 1/2	1.100	1.4219	0.600	6 7/16	3/64	3/32	1 3/16	57/64	1
12	10 1/8	1.200	1.5500	0.600	7	3/64	3/32	1 5/16	63/64	1 1/16
13	10 3/4	1.300	1.6750	0.600	7 1/2	3/64	7/64	1 7/16	1 5/64	1 1/16
14	11 3/8	1.400	1.8000	0.600	8	3/64	7/64	1 1/2	1 1/8	1 1/8
15	12	1.500	1.9281	0.600	8 9/16	3/64	7/64	1 5/8	1 7/32	1 1/8
16	12 5/8	1.600	2.0531	0.600	9 1/16	3/64	7/64	1 3/4	1 5/16	1 1/8
17	13 3/8	1.700	2.1812	0.600	9 5/8	3/64	7/64	1 13/16	1 23/64	1 1/4
18	14	1.800	2.3062	0.600	10 1/8	1/16	1/8	1 7/8	1 13/32	1 1/2
19	14 5/8	1.900	2.4312	0.600	10 5/8	1/16	1/8	1 15/16	1 29/64	1 1/4
20	15 1/4	2.000	2.5562	0.600	11 1/8	1/16	1/8	2	1 1/2	1 3/8
BROWN & SHARPE										
1	4 3/4	0.200	0.317	0.5000	2 7/8	1/40	1/32	1/4	1/4	3/8
2	5 1/8	0.250	0.377	0.5000	3 1/8	1/40	1/32	5/16	1/4	3/8
3	5 1/2	0.3125	0.450	0.5000	3 3/8	1/40	1/32	3/8	5/16	7/16
4	5 7/8	0.350	0.501	0.5000	3 11/16	1/40	1/32	7/16	3/8	1/2
5	6 3/8	0.450	0.614	0.5000	4	1/40	3/64	9/16	7/16	1/2
6	6 7/8	0.500	0.680	0.5000	4 3/8	1/40	3/64	5/8	1/2	9/16
7	7 1/2	0.600	0.800	0.5000	4 7/8	1/32	1/16	3/4	5/8	11/16
8	8 1/8	0.750	0.976	0.5000	5 1/2	1/32	5/64	7/8	11/16	3/4
9	8 7/8	0.900	1.152	0.5000	6 1/8	1/32	5/64	1 1/16	7/8	7/8
10	9 3/4	1.0446	1.337	0.5161	6 7/8	3/64	3/32	1 3/16	1	1
11	10 5/8	1.250	1.565	0.5000	7 5/8	3/64	3/32	1.45	1 1/8	1 1/8
12	11 3/8	1.500	1.841	0.5000	8 1/4	3/64	7/64	1.45	1 1/8	1 1/8
13	12	1.750	2.111	0.5000	8 3/4	3/64	7/64	1.62	1 1/4	1 1/4
14	12 1/2	2.000	2.382	0.5000	9 1/4	3/64	1/8	1.62	1 1/4	1 1/4
15	13 1/8	2.250	2.654	0.5000	9 3/4	3/64	1/8	1.95	1 1/2	1 3/8
16	13 1/2	2.500	2.924	0.5000	10 1/4	1/16	9/64	1.95	1 1/2	1 3/8
17	13 3/4	2.750	3.195	0.5000	10 3/4	1/16	9/64	1.95	1 1/2	1 3/8
18	14 1/4	3.000	3.466	0.5000	11 1/4	1/16	9/64	1.95	1 1/2	1 3/8

DRILLING*

DRILLS.—A drill is an end-cutting tool, consisting usually of two cutting edges set at an angle with the axis. The more common types of drills are flat—flat-twisted—straight fluted—spiral-fluted—and gun-barrel. The most common, and for most purposes the most efficient type, is the spiral-fluted, known as a twist drill.

Twist drills are made with two, three, or four cutting lips. The four-lip drills are used for enlarging holes previously cored or drilled. When drilling solid stock with a two-lipped drill, the point of the drill controls the cutting edges, and if the drill is correctly ground the resulting hole will be reasonably round, straight, and the size of the drill. When a drill is used for enlarging holes already made, either by coring or by previous drilling, the drill is guided by its sides and a three or four fluted drill will give better results.

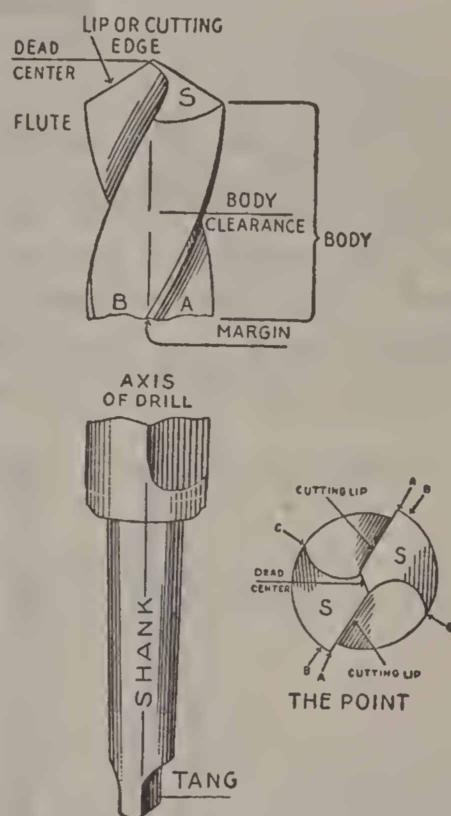


Fig. 41—Drill Parts

There are three principal parts to a twist drill—the point, the body and the shank, as shown in Fig. 41.

The point is the entire cone-shaped surface at the cutting end of the tool. The two or more spiral grooves running along the sides of the body of the drill are called flutes and serve four purposes. They help form the cutting edges of the point; they curl the chip so that it occupies the minimum space; they provide a means for the chip to escape from the drill hole; and they serve to convey the lubricant or cutting oil to the

*See pp. 47, 63, 97, 174 and 175, Vol. I, Starrett Books. Also pp. 10–12, Vol. II, Starrett Books.

cutting edges of the tool. The shank provides a means of holding the drill in the drill press.

Referring again to Fig. 41, it will be seen that portions of each of the three principal parts of the drill have names. The tip of point, or cone-shaped section, is called the dead center, and if the drill is to cut properly, this dead center must exactly coincide with the axis of the drill. The edges

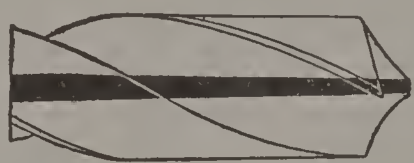


Fig. 42—Web of Drill (Shown by Dark Section)

of the point which do the actual cutting are called the lips and the portion of the point back of the lips is known as the heel. The narrow strip (A–B, Fig. 41) running along one edge of each flute is called the margin. It forms the outside cutting edge of the drill body and is the full diameter of the

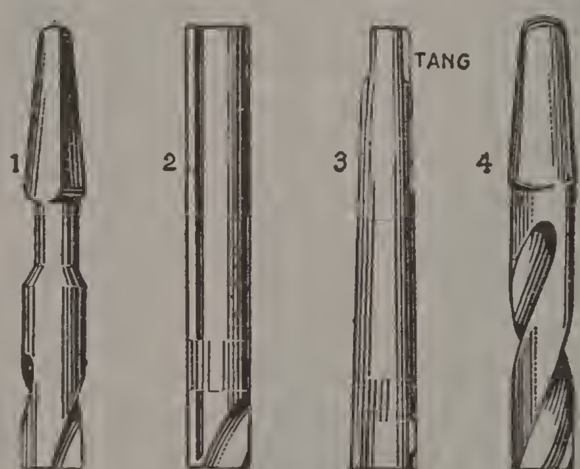


Fig. 43—Four Types of Drill Shank
(1) Bit Stock (2) Straight
(3) Taper (4) Ratchet

drill. At point B (Fig. 41), the drill is relieved by what is called body clearance. The web, as shown in Fig. 42, extends along the central axis of the drill and runs the entire

length of the body. It thickens toward the shank of the tool and gives the drill the necessary stiffness and strength.

The four common types of shanks used on drills are shown in Fig. 43. They are, from left to right, the bit stock shank, the straight shank, the taper shank and the ratchet shank. Straight and taper shanks are the types most commonly used in automobile work.

FORM OF POINT.—All drills used in ordinary machine shop practice, except gun-barrel drills, are cone-pointed on the cutting end. The gun-barrel drill, used when especially straight, round, and true holes are essential, has a blunt end with a single cutting lip.

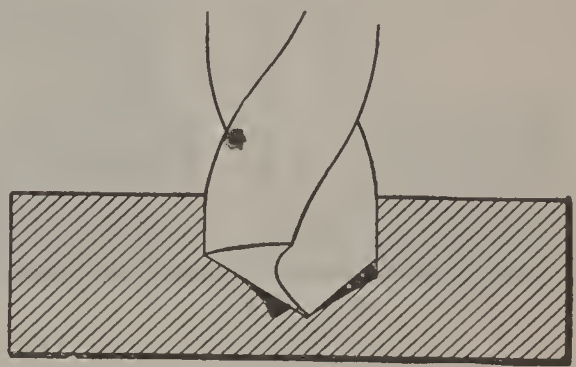


FIG. 44—TWIST DRILL—POINT INCORRECTLY GROUND

A cone-pointed drill of two or more cutting lips depends for its efficient working upon four factors:

(*a*) All the cutting lips shall have the same inclination to the axis of the drill.
(*b*) Cutting lips should be of exactly equal length.
(*c*) A proper lip clearance of the surface back of the cutting edges.
(*d*) A correct angle of lip clearance.

Figures 44, 45 and 46 show the result of careless free-hand grinding. Figs. 47 and 48 show one method of testing the length of the cutting lips, also their inclination to the axis. A better method, however, is to make use of a drill point gage (See Fig. 54), which checks at a single operation both the length of the lips and their angles with the axis of the drill.

After sharpening a drill free-hand, use the hand-feed at first and observe (*a*) the chips made by the cutting; (*b*) the size of the hole. If the cutting lips are shaped to a proper

clearance, the chips will curl as they start from the cutting edge. If the cutting lips lack a proper clearance, the resulting chips have the appearance of being ground off rather than freely cut. If the cutting lips are of uneven length the hole will be enlarged over the diameter of the drill. Drillings from cast iron should look as in Fig. 57, and those from steel as in Fig. 58, if the drill is properly sharpened.

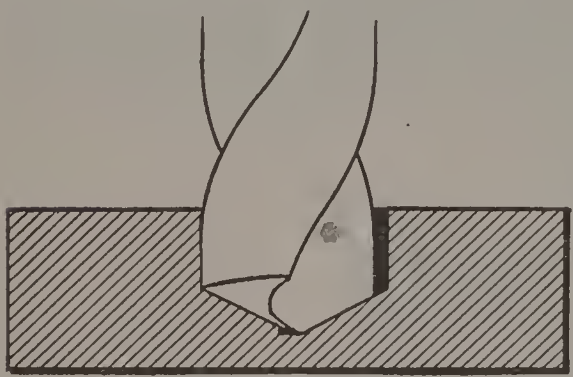


Fig. 45

Free-hand grinding results are usually so disappointing that in most machine shops the drills are sharpened in a special drill-grinding machine. The design of this machine is such that when it is set for grinding any size of drill the cutting lips are made of equal length and of the correct form. Fig. 59 shows how the cutting lip is located to grind the edges correctly. When grinding drills, care must be taken to avoid drawing the temper of the tool.

It may be safely and conservatively stated that 90 per cent of all drill troubles are the result of faulty grinding of either point or lip clearance. As was shown in Fig. 41, the lips are the cutting edges of the drill and are formed by grinding the point at such an angle that the dead center coincides exactly with the axis of the drill—thus making the lips of precisely equal length—and then grinding away, or backing off or relieving the heel, as is shown at the right of Fig. 47. Note the difference between the drill point at the right of Fig. 47 and that shown at the left. The drill at the left has

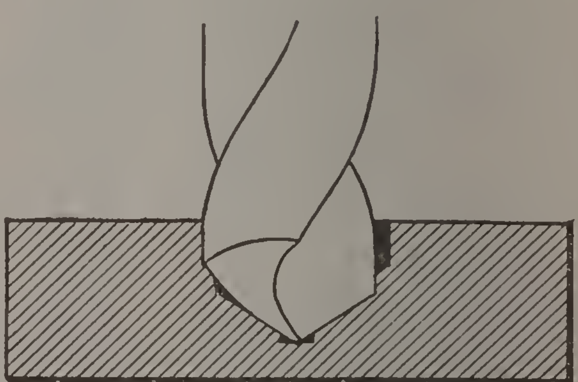


Fig. 46

been ground so that the cone-shaped surface is at right angles to the axis of the drill and as a result has no cutting edge. In order that the drill may cut, or enter the metal, the sur-

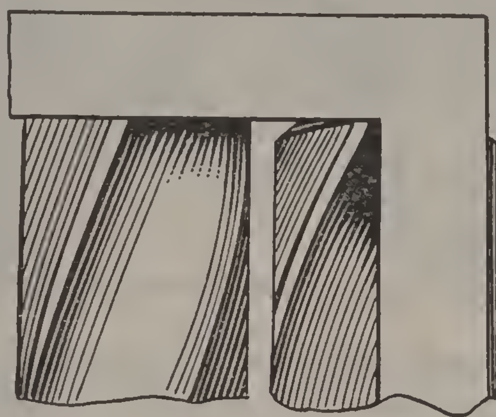


FIG. 47—CORRECT (RIGHT) AND INCORRECT (LEFT) GRINDING POINT CLEARANCE

faces S, see Figs. 48 and 49, must be ground away, so that the heel line B is below the cutting lip A, as is shown in Fig. 49.

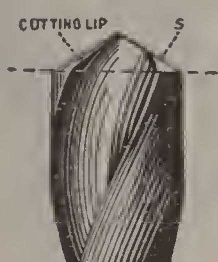


FIG. 48—DRILL GROUND WITHOUT LIP CLEARANCE

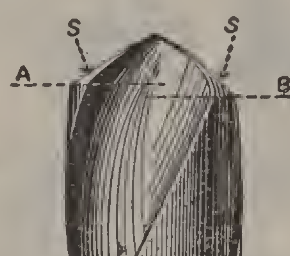


FIG. 49—DRILL GROUND WITH PROPER LIP CLEARANCE

Figure 50 shows the angle—12° to 5°—at which heels should be relieved from the cutting edge. The angle is measured at the circumference or outer edge of the drill and should increase as the center of the drill is approached.

When a properly ground drill is working in a hole, its condition will be similar to that shown in Fig. 51, where the

FIG. 50—PROPER ANGLE FOR GRINDING LIP CLEARANCE

cutting lip has already removed considerable metal in advance of the heel, as is indicated by the dark sections on either side of the drill.

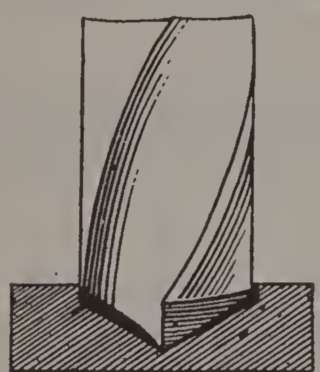


FIG. 51—CORRECTLY GROUND DRILL WORKING IN HOLE

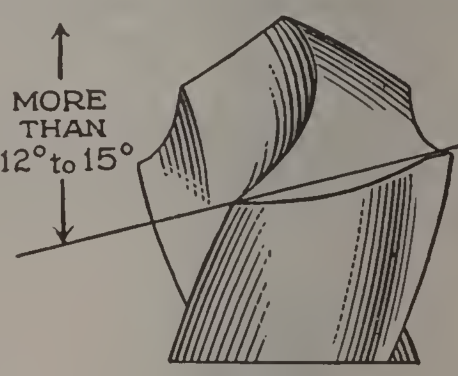


FIG. 52—TOO MUCH LIP CLEARANCE

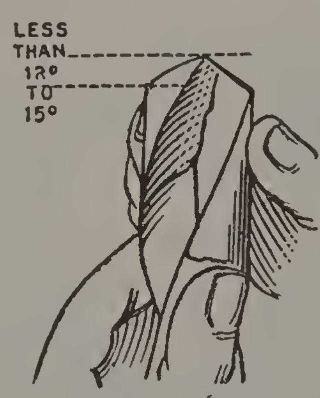


FIG. 53—TOO LITTLE LIP CLEARANCE

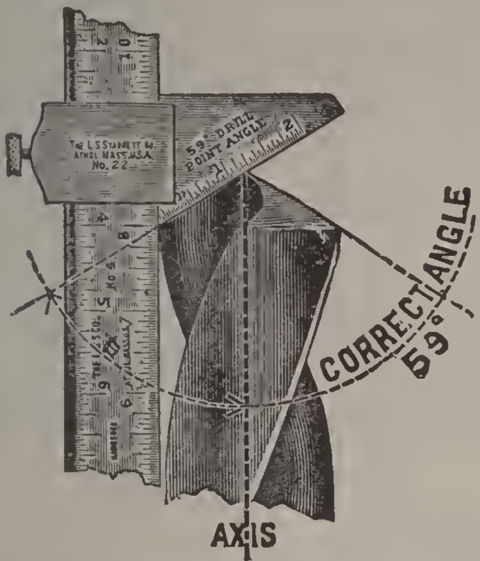


FIG. 54—DRILL GROUND WITH CORRECT LIP ANGLES

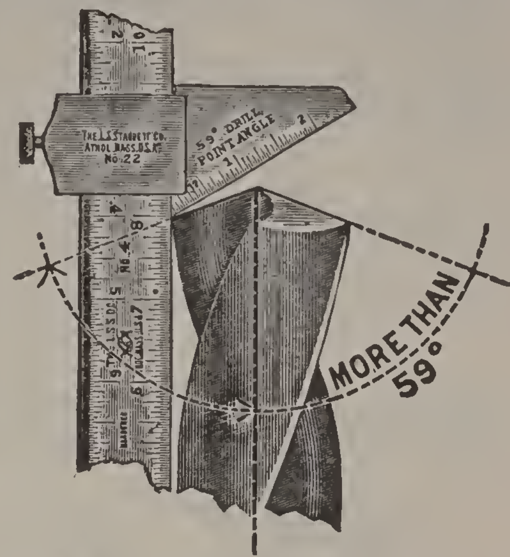


FIG. 55—DRILL GROUND WITH TOO FLAT A POINT

When too much lip clearance—over 12° to 15°—is given a drill point, as is shown in Fig. 52, the cutting edges are weakened so that they cannot withstand the pressure of drilling and will chip off or burn away.

Insufficient lip clearance—less than 12° to 15°—makes it difficult for the drill to enter the metal and often causes splitting of the drill as shown in Fig. 53.

A properly ground drill has its lips exactly the same in length and the angle the cutting edge makes with the axis of the drill is 59° as shown in Fig. 54.

If the angle the cutting edge makes with the axis of the drill is more than 59°—even though the lips of the drill are equal in length—the tool will not center properly, because the cone-shaped point which is intended to hold it in its central position will be too nearly flat, See Fig. 55. If the angle is less than 59° — see Fig. 56 — the

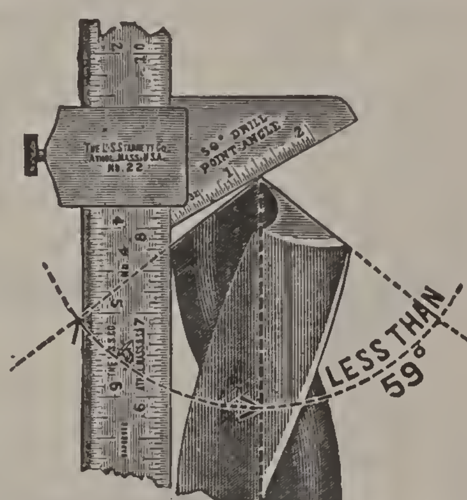


Figure 3

FIG. 56—DRILL GROUND WITH POINT TOO ACUTE

FIG. 57

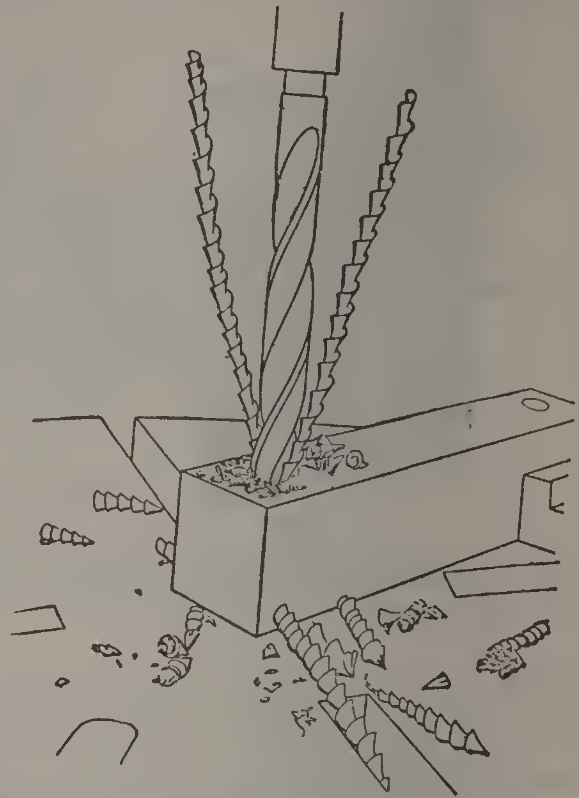


FIG. 58

drill will require more driving power because of the increased length of the cutting edges.

If the cutting edges or lips have the same and correct angle, but are of different length, the point will necessarily be off center and the hole will be larger than the drill and irregularly shaped. Incidentally, a drill improperly ground not only destroys both itself—in addition to doing poor work—but is also destructive of the drill press because of the weaving of the spindle caused by the point of the drill not being held in alignment with the spindle of the drill press.

FIG. 59

CUTTING COMPOUNDS*. To maintain high cutting speeds, it is necessary to use a lubricant. Those recommended have stood the test of service:

For hard and refractory steel—Turpentine, kerosene, or soda water.
For soft steel and wrought iron—Lard oil or soda water.
For brass—Paraffine oil.
For aluminum—Turpentine, kerosene, or soda water.
For cast iron—A jet of air if anything is used—usually worked dry.

LAYING OUT

Locating the centers for drilled holes upon the body of the work is termed "laying out". On the smaller jobs, laying out and drilling are usually done by the workman. Larger amounts of work warrant a skilled "layer out".

Laying out for drilling comes under two heads, viz.: APPROXIMATE and ACCURATE. Unless the holes when drilled are to match up with other holes or with fixed studs, it is usually sufficiently accurate if the center is laid off with a chalk pencil and a steel rule. For jig, tool, and experimental work, the centers must be accurately laid out and scribed upon the surface of the work. The practice is to scribe two

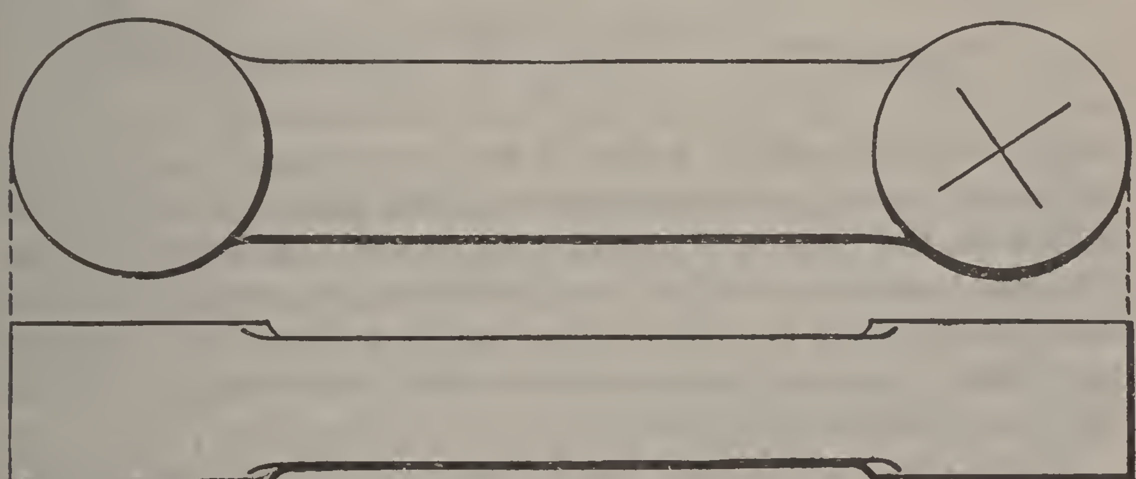


FIG. 60

*For cutting lubricants used in various operations see also p. 10, Vol. II, Starrett Books.

or more lines which intersect at the exact desired point as shown in Fig. 60. Assume that the link is to connect two studs. Proceed to scribe two intersecting lines upon one of the hubs, as shown in Fig. 60, using a combination square fitted with a center head. At the intersection, accurately place a light center-punch indentation. Place one leg of a

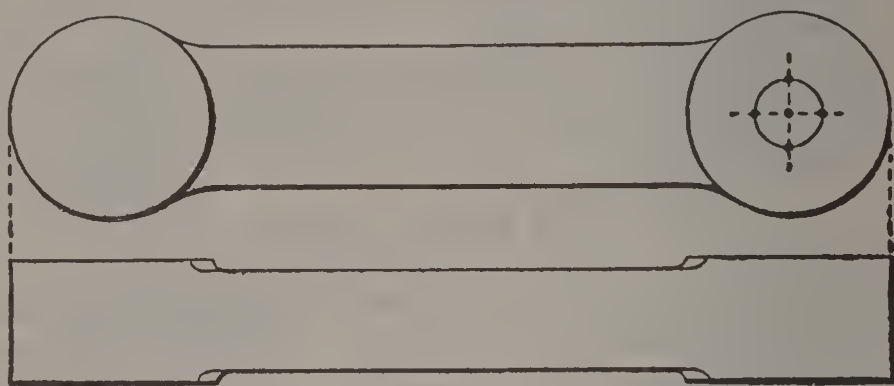


Fig. 61

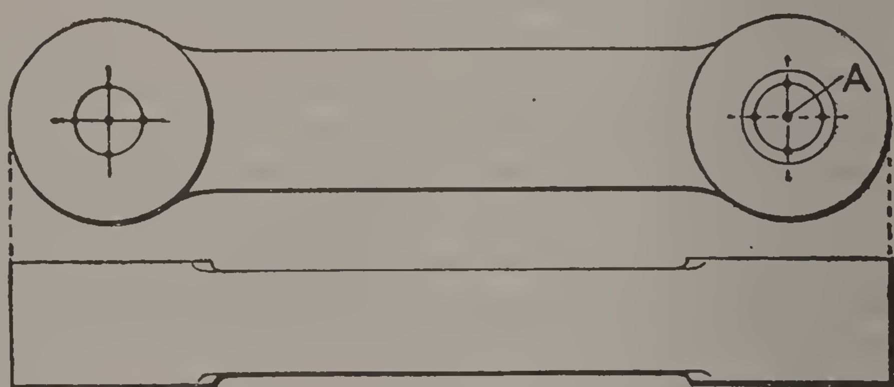


Fig. 62

spring divider with its point in the center mark and adjust the other leg to have its point touch the edge line of the hub and note the concentricity of the center. If correct, close dividers to scribe a circle the diameter of the required drilled hole, setting the points by the scale graduations upon a steel rule. Locate light center-punch marks on the scribed circle as shown in Fig. 61.

When the work is laid out by another than the driller, a second circle, having a slightly greater diameter, should be scribed. This check will show whether the hole was drilled

to the original layout. If no importance is attached to the center to center distance of the holes proceed as before with the second hub. Where the center to center distance is important, set the points of the universal dividers to the center length, and with the point A, Fig. 62, in the previously located center mark scribe on the opposite hub. Scribe a short line across its face afterward, proceeding as before.

For accurate work, the use of the automatic center-punch

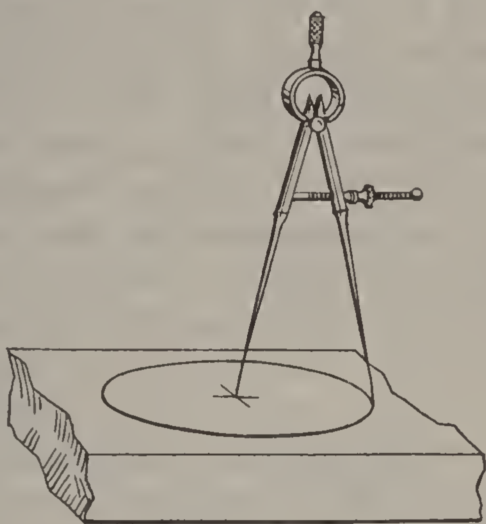


FIG. 63—SCRIBING CIRCLES WITH DIVIDERS

(Fig. 64) is recommended, as is the machinists' center-punch (shown in Fig. 65), for heavy work.

PREPARING THE SURFACE. For accurate laying out, clean the machined surfaces and wet the portion to be worked upon with a copper sulphate (blue vitriol) solution. To prepare a copper sulphate solution of the proper strength, dissolve one ounce of copper sulphate (commonly called blue vitriol) in four ounces of water to which has been added a scant teaspoonful of nitric acid. When dry, the surface as treated will distinctly show any lines which are made upon it. Chalk, well rubbed into the surface, is sufficient for the less accurate jobs.

STARTING THE DRILL. After laying out and previous to drilling, enlarge the center holes with a center-punch

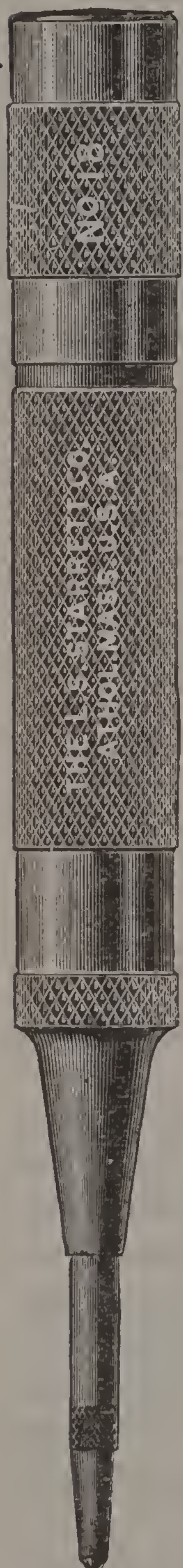


Fig. 64

to assist the starting of the drill. Start the hole with drill point in the enlarged center, using hand-feed until the nose of the drill is well started in the work. Observe if this is central with the scribed circle, and if not central use center gouge, as in Fig. 66, and repeat until accurate.

TO DRAW A DRILL. When starting a drill it often has a tendency to slide or crowd off to one side. Where it is essential that the drilled hole coincide or center with some previously scribed circle or layout, the drill must be brought back into the correct position. This is accomplished by the use of a small gouge-pointed chisel, sometimes called a center chisel, and the process is termed "drawing the drill". First, note toward which side of the small dimple left by the drill point it is necessary to shift the drill. Then chisel a small groove in that side of the dimple.

If the start is very eccentric, several chisel grooves may be necessary; whereas, if only slightly eccentric, a mere touch of the chisel will often suffice. It is readily seen that the drill is made to cut more easily where the grooves are, and therefore the natural resistance of the opposite side pushes the drill toward the side cut by the gouge-pointed chisel. Drill drawing can only be done previous to reaching the full diameter of cut.

HOLDING THE WORK. Carelessness in holding the work is responsible for many drilling accidents. If no special

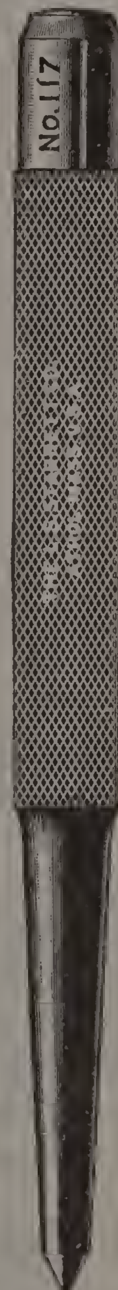


Fig. 65

holding device is available, the work should be held in a drilling vise, clamped directly to the drilling-machine table, or clamped to an angle iron. Fig. 67 illustrates a method of holding the work safely. When once the work is clamped in position on the drilling-machine table, adjust the table to center the located hole with the drill rather than reclamp the work. When drilling on a speed-lathe the same caution should be taken. Do not hold work in the hand, use a vise or the steady rest.

HOLDING THE DRILL. In Fig. 68, at A, the drill is shown held directly in the spindle. This is a good method if several holes of the same diameter are to be drilled at a single setting. When frequent changing of the drill is necessary, as in drilling holes of various different sizes, some form of quick-acting collet chuck should be used with single-spindle machines. The changes can then be made *without stopping the machine.*

DRILL PRESS

A common type of back-geared, upright drill press is shown on page 59, the various parts and their functions being indicated by the notations on the drawing.

USING TWIST DRILLS

In using twist drills too much attention can scarcely be given to speed and feed of the drill. Speed, ordinarily, does not refer to revolutions per minute, but to the peripheral or circumferential speed of the drill. In other words, the distance the drill would travel if it were laid on its side and rolled. Thus, a drill speed of 30 feet per minute means that the tool—traveling on its side—would roll 30 feet in one minute. Feed is the distance per minute the drill advances in the work and feed pressure is the pressure required to maintain this rate of advance.

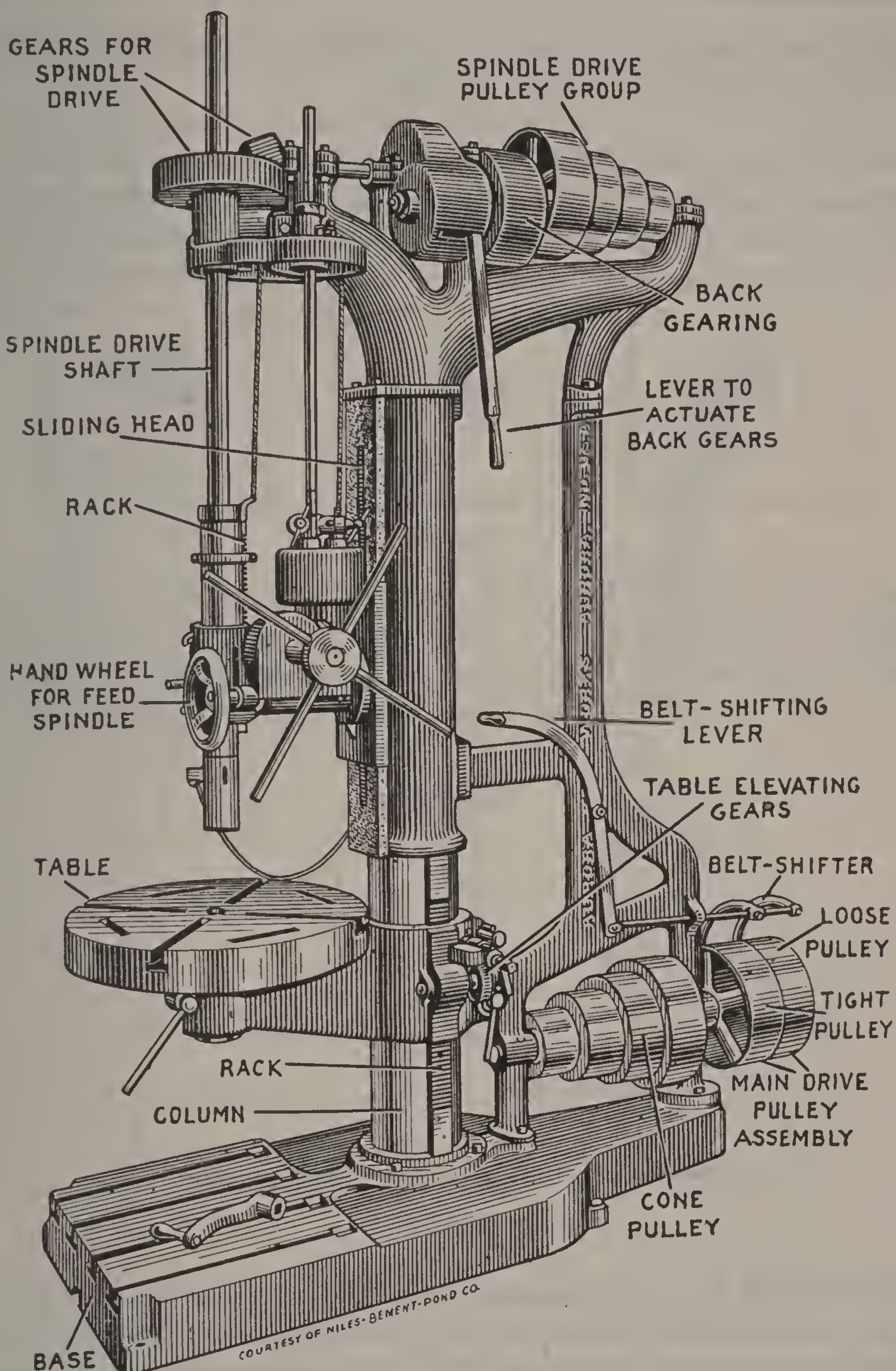
The correct feeds and speeds to use in drilling vary widely with the character of the metal being drilled, the character

SPEEDS AND FEEDS FOR DRILLING, USING CARBON STEEL DRILLS

(High-Speed Steel Drills run 100% faster)

Material		Bronze Brass		Annealed Cast Iron		Hard Cast Iron *a*		Mild Steel *b*		Malleable Iron	
Cutting Speed		150′ per min.		85′ per min.		40′ per min.		60′ per min.		45′ per min.	
Size of Drill	Feed per Rev.	Rev. per min.	Feed per min.	Rev. per min.	Feed per min.	Rev. per min.	Feed per min.	Rev. per min.	Feed per min.	Rev. per min.	Feed per min.
Ins.	Ins.		Ins.		Ins.		Ins.		Ins.		Ins.
1/16	.003	9167	27.5	5195	15.59	2445	7.33	3667	11.00	2750	8.25
1/8	.004	4584	18.3	2597	10.39	1222	4.89	1833	7.33	1375	5.50
3/16	.005	3056	15.3	1732	8.66	815	4.07	1222	6.11	917	4.51
1/4	.006	2292	13.8	1299	7.79	611	3.67	917	5.50	688	4.13
5/16	.007	1833	12.8	1039	7.27	489	3.42	733	5.13	550	3.85
3/8	.008	1528	12.2	866	6.93	407	3.26	611	4.89	458	3.67
7/16	.009	1310	11.8	742	6.68	349	3.14	524	4.72	393	3.54
1/2	.010	1146	11.5	649	6.49	306	3.06	458	4.58	344	3.44
5/8	.011	917	10.1	519	5.71	244	2.69	367	4.03	275	3.03
3/4	.012	764	9.2	433	5.20	204	2.45	306	3.67	229	2.75
7/8	.013	655	8.5	371	4.82	175	2.27	262	3.41	196	2.55
1	.014	573	8.0	325	4.55	153	2.14	229	3.21	172	2.41
1 1/4	.016	458	7.3	260	4.16	122	1.96	183	2.93	138	2.21
1 1/2	.016	382	6.1	216	3.46	102	1.63	153	2.44	115	1.83
1 3/4	.016	327	5.23	186	2.99	87	1.40	131	2.09	98	1.57
2	.016	286	4.58	162	2.60	76	1.22	115	1.83	86	1.38
2 1/4	.016	255	4.07	144	2.30	68	1.08	102	1.63	76	1.22
2 1/2	.016	229	3.66	130	2.08	61	0.98	92	1.47	69	1.11
2 3/4	.016	208	3.33	118	1.89	55	0.88	84	1.34	63	1.00
3	.016	191	3.05	108	1.73	51	0.81	76	1.22	57	0.92

a. For cast steel, use half the speed shown for hard cast iron.
b. For tool steel or drop forgings, use half the speed shown for mild steel.
The figures above can be attained under ideal conditions, which cannot always be obtained in the shop. They form a mark at which to aim and thus show how far local shop conditions fall below the ideal.



BACK-GEARED, UPRIGHT DRILLING MACHINE

of the steel from which the drill itself is made and the temper of the particular drill being used.

Twist drills are divided into two classes according to the kind of steel from which they are made—carbon steel twist drills and high speed twist drills. The latter are more expensive, but can be operated at speeds greatly in excess of those

FIG. 66

permissible with carbon drills, and will do a correspondingly greater amount of work. High speed drills also hold their size better, require less sharpening and do more drilling in the same length of time than carbon drills. High speed

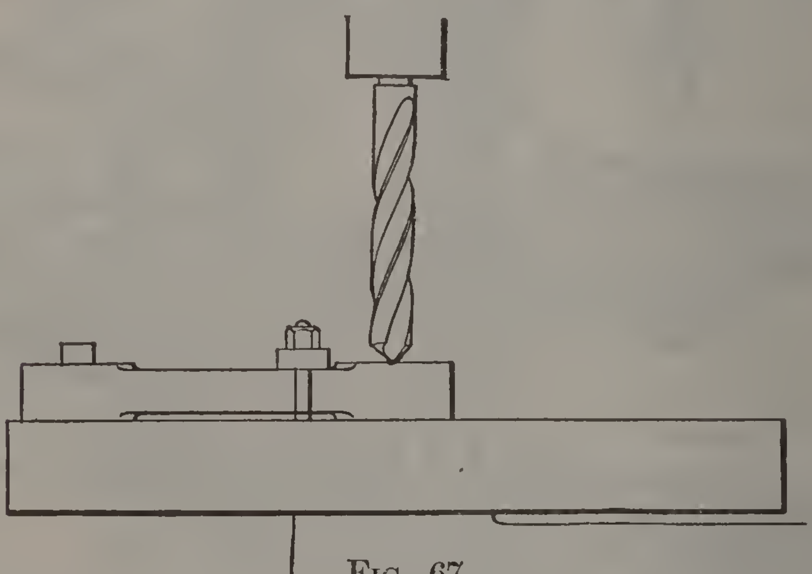
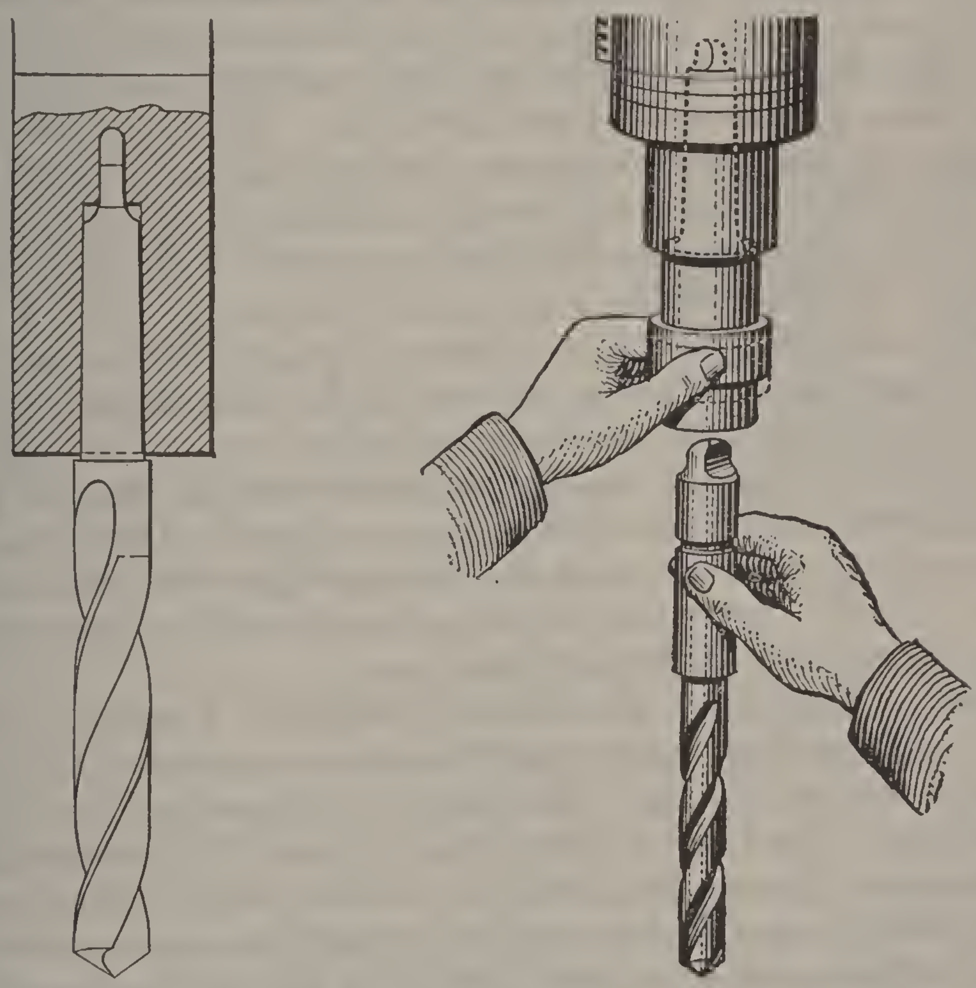


FIG. 67

drills should be used wherever possible, especially in all portable electric drills, because the spindles of this type of drill usually run at speeds far too high for carbon drills of ¼ inch diameter and larger.

SELECTING SPEEDS AND FEEDS FOR DRILLS

Common sense, experience and judgment are really the big factors in determining the correct drill feed and speed in each individual case. There are no hard and fast rules that can be invariably followed. Certain generalities and precautions may, however, be observed to advantage.



A

FIG. 68

When starting a drilling operation in a power drill press, always bring the drill down to the work by hand feed until it is centered in the work. Then only can the power feed be safely thrown in.

When drilling soft tool or machinery steel with carbon steel drills it is safe to start with a speed of 30 feet per minute. In cast iron a speed of 35 feet and in brass 60 feet per minute may be safely used in starting the work. If high speed drills are used these speeds may be doubled.

Recommended starting feeds vary according to the size of the drill and are the same for both high speed and carbon steel drills. For drills from 3/16 to 1/2 in. a feed of from .004 to .007 in. per rotation is recommended, while for drills over 1/2 in. in diameter the feed should be .005 to .015 inch per revolution of the drill. Drills under 3/16 inch take even lighter feeds than .004 inch. The secret of rapid drilling, as will be shown later, lies in using a light feed and the maximum permissible speed. This explains in part why more work can be done in the same time with high speed than with carbon drills.

When starting a drill it is wise to use moderate feed and speed and increase either or both only after observing the condition of the drill, character of chips, etc. If a properly ground drill chips at the cutting edge or splits up the web, it is a pretty sure indication of excessive feed. Rapid wearing away at the outer corner of the cutting edge is evidence of too much speed. Attention should also be paid to the proper use of lubricant. (See page 53.)

DRILLING FOR REAMER. When it is essential that the holes be of an exact diameter, it is customary to use a drill somewhat smaller than the given diameter, and afterward ream the holes to the desired size. The amount left for reaming depends upon whether one or two reaming operations are necessary, and whether or not the reaming is to be done with a power reamer directly in the drilling machine. If the drilling is done through jig bushings and the holes are short as compared to their diameter, a single reaming operation will often suffice. If the holes are relatively long, the drill should be 1/64 in. to 1/32 in. smaller than the finished hole diameter, to allow for passing a machine reamer

.005 in. small through the hole which is afterward hand reamed. This method gives results as accurate as any, except by grinding, and is accepted practice for good work.

DRILLING FOR TAPPING. Where a full thread depth is essential the hole to be tapped should be made with a drill of a diameter smaller than the nominal diameter of the bolt by an amount equal to double the depth of the thread. In practice the nearest commercial size of drill is listed for drilling tapped holes. The judgment of the operator, however, is the surest and safest guide in determining the proper size of drill to use.

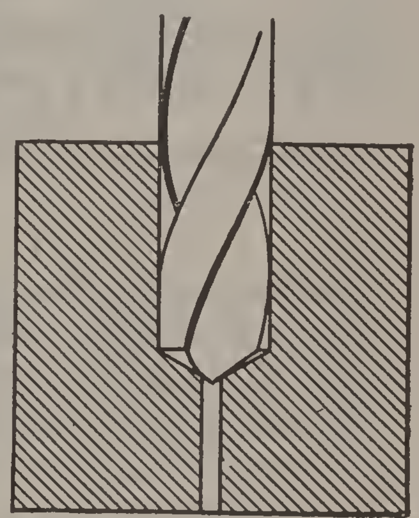


FIG. 69

DRILLING LARGE HOLES. Twist drills range in size from No. 80 wire gage to four inches in diameter. As the drill increases in diameter the web is correspondingly thickened, and as the cutting edges at the web do not cut as effectively as they do outside the web thickness, considerable pressure is required to force the larger drills into the work at an efficient speed. For this reason many workmen first drill a lead hole, using a drill whose diameter approximates the web thickness of the larger drill, as shown in Fig. 69. A lead hole will also assist in centering the drill upon an inclined surface. However, if the inclination is considerable it is necessary to butt mill or hand chip a spot giving sufficient surface to work upon. The practice of some firms engaged in production work, manufacturing, etc., is to use—in place of a single large drill—a relatively smaller one, afterward enlarging the hole by some method of counterboring at a much less expense for tools and at as rapid a production rate as by drilling the entire job.

BOLT HOLES. When the bolts are for holding purposes

only and are not used for aligning the several pieces, it is customary to drill the holes through which the bolts pass somewhat larger than the bolt diameters. This allows for a variation in the bolt sizes and for inaccuracy in locating the centers.

DEEP HOLE DRILLING. Under this name may be classed the drilling of holes through the axes of spindles, cam and crankshaft, push-rods, etc. While for spindle drilling it is possible to use ordinary twist drills with extended shanks, it is customary in efficient drilling of this sort to use special

Fig. 70

drills designed for the purpose. Fig. 70 shows a special hollow drill often used for drilling axial holes in spindles, and Fig. 71 shows the machine with the drill guides in working position.

In all cases of deep hole drilling it is better to rotate the work rather than the drill. The drill must be started exactly concentric with the axis of the machine. For this reason a starting-hole the exact diameter of the drill is first counterbored.

COUNTERBORING. There are many cases in which it is desirable to enlarge a hole throughout a portion of its

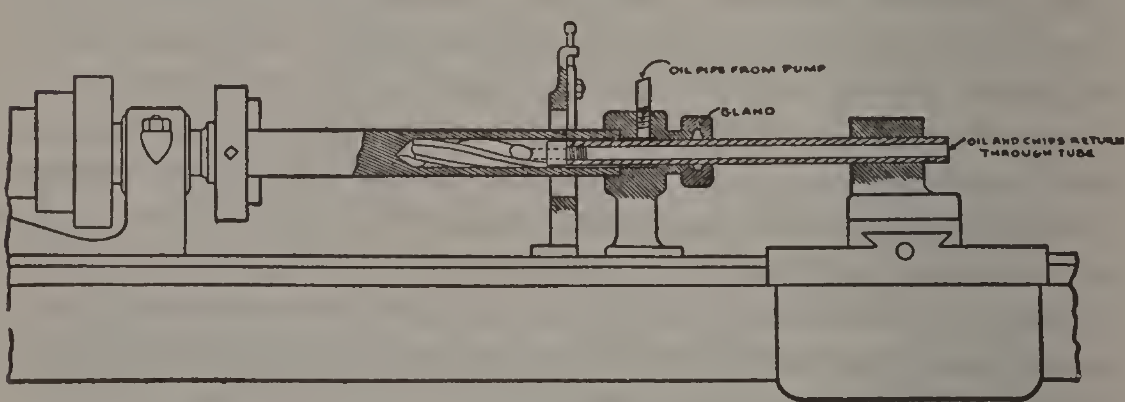


Fig. 71

length. If a drill is used for this purpose there is no certainty that the two diameters will be concentric. The practice is to enlarge the already drilled hole by using a cutting tool having a pilot or leader to guide the cutting edges. This tool is known as a counterbore, and its use is termed counterboring. Fig. 72 shows the tool in operation and its purpose.

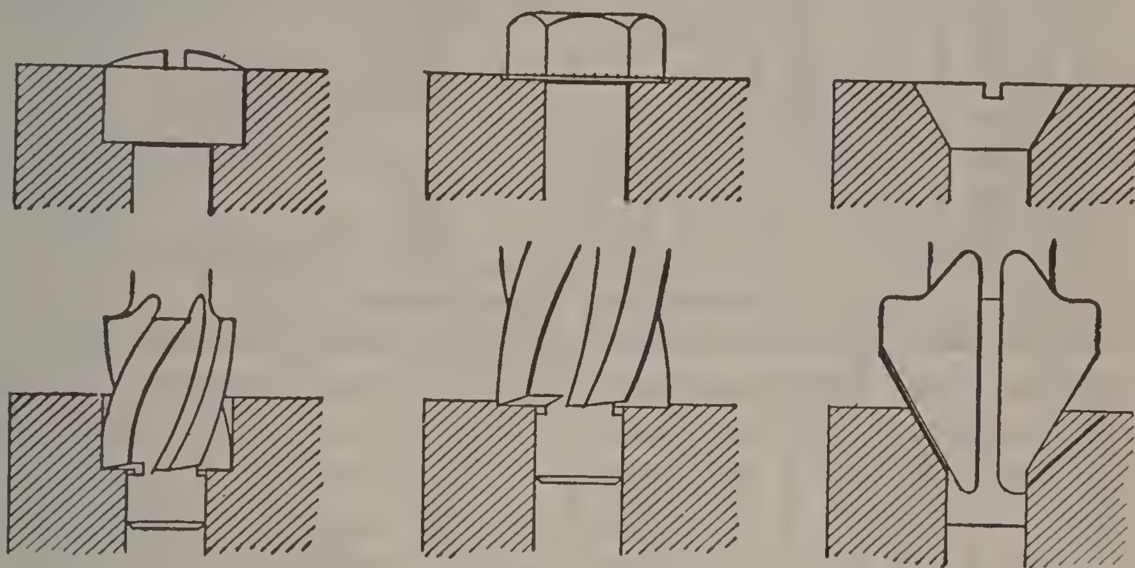


Fig. 72

NUMBERED TWIST DRILL SIZES

The usual drill set consists of numbers from 1 to 60. The smaller sizes are not so much used in automobile shops. Sets of alternate sizes, 1, 3, 5, etc., can also be bought. Drills down to about 40 have the numbers stamped on the shanks. The smaller ones have to be gaged.

LETTER SIZE TWIST DRILLS

The letter size twist drills are a continuation from the numbered sizes and are very useful for exact tap drill sizes in between the fractional sizes and to bridge the gap between the largest numbered size and the ¼ in. drill which is often the smallest fractional size drill stocked.

Numbered Twist Drill Sizes

No.	Diam.	No.	Diam.	No.	Diam.	No.	Diam.	No.	Diam.	No.	Diam.
1	.228	15	.180	28	.141	41	.0960	55	.0520	68	.0310
2	.221			29	.136	42	.0935			69	.0292
3	.213	16	.177	30	.129	43	.0980	56	.0465	70	.0280
4	.209	17	.173			44	.0860	57	.0430		
5	.206	18	.170			45	.0820	58	.0420	71	.0260
		19	.166	31	.120			59	.0410	72	.0250
6	.204	20	.161	32	.116	46	.0810	60	.0400	73	.0240
7	.201			33	.113	47	.0785			74	.0225
8	.199	21	.159	34	.111	48	.0760	61	.0390	75	.0210
9	.196	22	.157	35	.110	49	.0730	62	.0380		
10	.194	23	.154			50	.0700	63	.0370	76	.0200
		24	.152	36	.1065			64	.0360	77	.0180
11	.191	25	.150	37	.1040	51	.0670	65	.0350	78	.0160
12	.189			38	.1015	52	.0635			79	.0145
13	.185	26	.147	39	.0995	53	.0595	66	.0330	80	.0135
14	.182	27	.144	40	.0980	54	.0550	67	.0320		

Letter Size Twist Drills

Letter	Frac.	Deci.	Letter	Frac.	Deci.	Letter	Frac.	Deci.	Letter	Frac.	Deci.
A	15/64	.234	H	17/64	.266	O	5/16	.316	U	—	.368
B	—	.238	I	—	.272				V	3/8	.377
C	—	.242	J	—	.277	P	21/64	.323	W	25/64	.386
D	—	.246				Q	—	.332	X	—	.397
E	1/4	.250	K	9/32	.281	R	11/32	.339	Y	13/32	.404
			L	—	.290	S	—	.348	Z	—	.413
F	—	.257	M	19/64	.295	T	23/64	.358			
G	—	.261	N	—	.302						

DRILL SIZES FOR TAPS

Every motor mechanic knows that a hole must be drilled before a thread tap can be used. Many, however, do not realize the close relationship between the size of the drill and that of the tap which follows it, nor do they know the reason for the relationship. It is perfectly obvious that the drill must be slightly smaller in diameter than is the tap which cuts the thread in the hole, for if the hole was drilled to the exact size of the bolt or stud it is to take, there would be no metal left in which to cut the threads. The proper drill for each size of thread tap is called the tap drill and the difference in diameter between taps and tap drills is regulated by the character or kind of thread to be cut in each instance.

The explanation for this difference in the size of the tap drill used for the same sized tap in these threads is found in the effect of the type of thread to be cut and the difference in the number of threads to the inch. Fig. 73A shows the drill hole required for a 1¼ inch "V" thread bolt. Note that the outside diameter of the threads is 1¼ in., but that the drill hole required is but 1 3/64 inch. Compare this with the U. S. S. bolt as in Fig. 73B. Since the U. S. S. thread has a flattened top and root, to reduce the amount of metal to be removed by the tap, a larger drill is used. The space between the two inner sets of dotted lines indicates the difference in the amount of metal removed. Fig. 73C indicates in the same way the still larger drill required where the hole is to be threaded to take an S. A. E. bolt or machine screw. In this case, not only is the thread different from the V type, but the number of threads to the inch is 12, as compared with 7 for the V and U. S. S. types, making it even more necessary to remove a greater amount of metal before attempting to tap

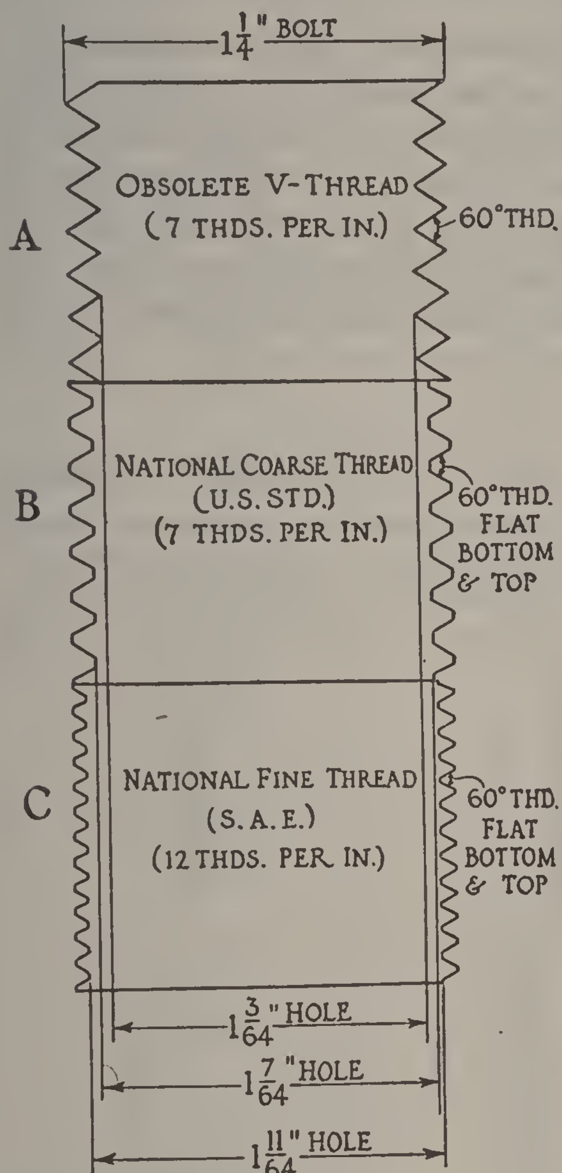


FIG. 73—COMPARISON OF TAP DRILL SIZES REQUIRED FOR BOLT HOLES OF SAME DIAMETER "V", U. S. S., AND S. A. E. THREADS

the hole. The different sizes of tap drills required for taps for "V", U. S. S., and S. A. E. Threads are given in the tables in this section.

When drilling holes to be threaded, some consideration should be given to the material on which the work is done and to the depth of the tapped hole. Generally speaking, soft, tough material, such as copper, Norway iron, drawn aluminum, etc., should have a larger hole drilled for the tap than is necessary when working in mild steel, cast iron, etc. Also, on soft, tough material, the sharpness of the tap is more important than with brass, steel, cast iron, etc.

COMPARISON OF TAP DRILL SIZES FOR DIFFERENT THREADS

Bolt Diameter	National Coarse Thread (Formerly U. S. Std.)	National Fine Thread (Formerly S. A. E.)
No. 1	No. 53	No. 53
2	51	50
3	47	46
4	44	42
5	39	38
6	36	33
8	29	29
10	26	21
12	17	15
1/4	8	3
5/16	1/4	9/32
3/8	5/16	11/32
7/16	23/64	25/64
1/2	27/64	29/64
9/16	31/64	33/64
5/8	17/32	37/64
3/4	21/32	11/16
7/8	49/64	13/16
1	7/8	15/16
1 1/8	63/64	1 3/64
1 1/4	1 7/64	1 11/64
1 1/2	1 21/64	1 27/64
1 3/4	1 35/64	1 43/64
2	1 25/32	1 59/64
2 1/4	2 1/32	2 5/32
2 1/2	2 1/4	2 13/32
2 3/4	2 1/2	2 21/32
3	2 3/4	2 57/64

Continuous Drill Table

Sizes starting with No. 80 and going up to 1 inch. This table is useful for quickly determining the nearest drill size for any decimal, for root diameters, body drills, etc.

Drill No.	Frac.	Deci.
80	—	.0135
79	—	.0145
—	1/64	.0156
78	—	.0160
77	—	.0180
76	—	.0200
75	—	.0210
74	—	.0225
73	—	.0240
72	—	.0250
71	—	.0260
70	—	.0280
69	—	.0292
68	—	.0310
—	1/32	.0313
67	—	.0320
66	—	.0330
65	—	.0350
64	—	.0360
63	—	.0370
62	—	.0380
61	—	.0390
60	—	.0400
59	—	.0410
58	—	.0420
57	—	.0430
56	—	.0465
—	3/64	.0469
55	—	.0520
54	—	.0550
53	—	.0595
—	1/16	.0625
52	—	.0635
51	—	.0670
50	—	.0700
49	—	.0730
48	—	.0760
—	5/64	.0781
47	—	.0785
46	—	.0810
45	—	.0820
44	—	.0860
43	—	.0890

Drill No.	Frac.	Deci.
42	—	.0935
—	3/32	.0938
41	—	.0960
40	—	.0980
39	—	.0995
38	—	.1015
37	—	.1040
36	—	.1065
—	7/64	.1094
35	—	.1100
34	—	.1110
33	—	.1130
32	—	.116
31	—	.120
—	1/8	.125
30	—	.129
29	—	.136
—	9/64	.140
28	—	.141
27	—	.144
26	—	.147
25	—	.150
24	—	.152
23	—	.154
—	5/32	.156
22	—	.157
21	—	.159
20	—	.161
19	—	.166
18	—	.170
—	11/64	.172
17	—	.173
16	—	.177
15	—	.180
14	—	.182
13	—	.185
—	3/16	.188
12	—	.189
11	—	.191
10	—	.194
9	—	.196
8	—	.199

Drill No.	Frac.	Deci.
7	—	.201
—	13/64	.203
6	—	.204
5	—	.206
4	—	.209
3	—	.213
—	7/32	.219
2	—	.221
1	—	.228
A	—	.234
—	15/64	.234
B	—	.238
C	—	.242
D	—	.246
—	1/4	.250
E	—	.250
F	—	.257
G	—	.261
—	17/64	.266
H	—	.266
I	—	.272
J	—	.277
—	9/32	.281
K	—	.281
L	—	.290
M	—	.295
—	19/64	.297
N	—	.302
—	5/16	.313
O	—	.316
P	—	.323
—	21/64	.328
Q	—	.332
R	—	.339
—	11/32	.344
S	—	.348
T	—	.358
—	23/64	.359
U	—	.368
—	3/8	.375
V	—	.377
W	—	.386
—	25/64	.391

Drill No.	Frac.	Deci.
X	—	.397
Y	—	.404
—	13/32	.406
Z	—	.413
—	27/64	.422
—	7/16	.438
—	29/64	.453
—	15/32	.469
—	31/64	.484
—	1/2	.500
—	33/64	.516
—	17/32	.531
—	35/64	.547
—	9/16	.562
—	37/64	.578
—	19/32	.594
—	39/64	.609
—	5/8	.625
—	41/64	.641
—	21/32	.656
—	43/64	.672
—	11/16	.688
—	45/64	.703
—	23/32	.719
—	47/64	.734
—	3/4	.750
—	49/64	.766
—	25/32	.781
—	51/64	.797
—	13/16	.813
—	53/64	.828
—	27/32	.844
—	55/64	.859
—	7/8	.875
—	57/64	.891
—	29/32	.906
—	59/64	.922
—	15/16	.938
—	61/64	.953
—	31/32	.969
—	63/64	.984
—	1	1.000

U. S. Standard Tap Drill Table

The tap drills listed here will give about a 75-80% thread which is sufficiently strong to break the bolt before stripping the threads. A full thread can be tapped by drilling to the root diameter, but the breakage of taps will be heavy and no good will be gained. For rough work a 50% thread is considered sufficient.

Diameter	Threads Per Inch	Root Diameter	Tap Drill
1/4	20	.185	No. 8
5/16	18	.240	1/4
3/8	16	.294	5/16
7/16	14	.345	23/64
1/2	13	.400	27/64
9/16	12	.454	31/64
5/8	11	.507	17/32
3/4	10	.620	21/32
7/8	9	.731	49/64
1	8	.838	7/8
1 1/8	7	.939	63/64
1 1/4	7	1.064	1 7/64
1 1/2	6	1.283	1 21/64
1 3/4	5	1.490	1 35/64
2	4 1/2	1.711	1 25/32
2 1/4	4 1/2	1.961	2 1/32
2 1/2	4	2.175	2 1/4
2 3/4	4	2.425	2 1/2
3	4	2.675	2 3/4

NOTE: For threads above 3 in., areas of bolts, areas at roots, head and nut dimensions, etc., see The Starrett Book, Volume I, Page 78.

SIZES OF TAP DRILLS FOR TAPS WITH U. S. S., S. A. E., "V", AND WHITWORTH THREADS

"V" Thread			U. S. S.			S. A. E.			Whitworth		
Diam. of Tap in inches	Threads per inch	Size of Drill No.	Diam. of Tap in inches	Threads per inch	Size of Drill No.	Diam. of Tap in inches	Threads per inch	Size of Drill No.	Diam. of Tap in inches	Threads per inch	Size of Drill No.
1/4	20	13	1/4	20	8	1/4	28	3	1/4	16	3/16
5/16	18	D	5/16	18	1/4	5/16	24	9/32	5/16	16	15/64
3/8	16	M	3/8	16	5/16	3/8	24	11/32	3/8	14	9/32
7/16	14	11/32	7/16	14	23/64	7/16	20	25/64	7/16	14	11/32
1/2	13	Y	1/2	13	27/64	1/2	20	29/64	1/2	12	3/8
9/16	12	29/64	9/16	12	31/64	9/16	18	33/64	—	—	—
5/8	11	1/2	5/8	11	17/32	5/8	18	37/64	5/8	10	1/2
3/4	10	39/64	3/4	10	21/32	3/4	16	11/16	3/4	10	5/8
7/8	9	23/32	7/8	9	49/64	7/8	14	13/16	7/8	9	23/32
1	8	53/64	1	8	7/8	1	14	15/16	1	8	27/32
1 1/8	7	15/16	1 1/8	7	63/64	1 1/8	12	1 3/64	—	—	—
1 1/4	7	1 1/16	1 1/4	7	1 7/64	1 1/4	12	1 11/64	—	—	—
1 1/2	6	1 9/32	1 1/2	6	1 21/64	1 1/2	12	1 27/64	—	—	—
1 3/4	5	1 1/2	1 3/4	5	1 35/64	1 3/4	12	1 43/64	—	—	—
2	4 1/2	1 23/32	2	4 1/2	1 25/32	2	12	1 59/64	—	—	—
2 1/4	4 1/2	1 31/32	2 1/4	4 1/2	2 1/32	2 1/4	12	2 5/32	—	—	—
2 1/2	4	2 11/64	2 1/2	4	2 1/4	2 1/2	12	2 13/32	—	—	—
2 3/4	4	2 27/64	2 3/4	4	2 1/2	2 3/4	12	2 21/32	—	—	—
3	4	2 43/64	3	4	2 3/4	3	10	2 57/64	—	—	—

NOTE: The drills recommended for various sizes and kinds of taps will leave enough metal to form approximately a 75-80 per cent thread. For exact sizes and percentages of threads obtained, see pages 97 and 98.

SCREW THREAD FITS

The particular use to which a screw or nut is put may make quite a difference in the fit of the screw threads. The extreme case of a sloppy fit is a stove bolt and nut, usually used to hold sheet metal parts, fenders, aprons and other parts together. Here the nut is a loose and wobbly fit, but it is sufficient for the purpose. The other extreme is in main bearing studs and nuts where the material must be of decidedly better grade than cold rolled steel and where the threads must be a very close fit so that the strain will be equally distributed between the different threads on the screw and nut.

The American standard of screw threads, in addition to recommending certain basic sizes and thread pitches as shown in the tables on pages 97 and 98, has made definite recommendations for various classes of fits as follows:

LOOSE FIT: Recommended as a commercial standard for tapped holes in the numbered sizes only. May be used with screws in other classes to obtain quality of fit desired.

FREE FIT: Includes the great bulk of screw thread work of ordinary quality of finished and semi-finished bolts, nuts, etc.

MEDIUM FIT: Includes the better grade of interchangeable screw thread work such as automobile bolts and nuts.

CLOSE FIT: Includes screw thread work requiring a fine snug fit, somewhat closer than the medium fit, such as high grade aircraft parts, etc. In this class of fit selective assembly of parts may be required. It is not considered practicable as a commercial standard for tapped holes of the numbered sizes.

It should be noted that the fit of a screw thread is controlled in the cutting of the thread and has no relation whatever to the diameter of the tap drill hole. Increasing the size of this hole simply takes off the crest of the nut thread, but in no way affects the closeness of fit of the rest of the thread.

TAP DRILLS

S. A. E. Screw Thread Table

Adopted by the Society of Automotive Engineers, June, 1911. Previously known as the A. L. A. M. thread. Intended for use in steel and hard materials. Cast iron, bronze and aluminum should be threaded U. S. Std. The tap drills listed allow about 75% thread which is sufficiently strong to break the bolt before stripping the thread.

Diameter	Threads per Inch	Tap Drill	Diameter	Threads per Inch	Tap Drill
1/4	28	No. 3	3/4	16	11/16
5/16	24	9/32	7/8	14	13/16
3/8	24	11/32	1	14	15/16
7/16	20	25/64	1 1/8	12	1 1/64
1/2	20	29/64	1 1/4	12	1 9/64
9/16	18	33/64	1 1/2	12	1 27/64
5/8	18	37/64			

Above 1 1/2 in. the S. A. E. threads have two pitches, coarse and fine.

Diameter	Coarse	Fine	Diameter	Coarse	Fine
1 5/8	12	16	3 7/8	10	16
1 3/4	12	16	4	10	16
1 7/8	12	16	4 1/8	10	16
2	12	16	4 1/4	10	16
2 1/8	12	16	4 3/8	10	16
2 1/4	12	16	4 1/2	10	16
2 3/8	12	16	4 5/8	10	16
2 1/2	12	16	4 3/4	10	16
2 5/8	12	16	4 7/8	10	16
2 3/4	12	16	5	10	16
2 7/8	12	16	5 1/8	10	16
3	10	16	5 1/4	10	16
3 1/8	10	16	5 3/8	10	16
3 1/4	10	16	5 1/2	10	16
3 3/8	10	16	5 5/8	10	16
3 1/2	10	16	5 3/4	10	16
3 5/8	10	16	5 7/8	10	16
3 3/4	10	16	6	8	16

Sizes from 6 in. up have a coarse pitch of 8 and a fine pitch of 16. Sizes run by eighths.

TAP AND BODY DRILLS

Machine Screw Threads*

The body drills listed are exact sizes of the screw. Where more clearance is desired use one or two sizes larger. The tap drills give a 75% thread which is sufficient. For steel, use one or two sizes larger. Threads marked (*) are National Fine Thread Series (S. A. E. Standard); those marked (†) are National Coarse Series (U. S. Standard) (See page 85).

Tap	Diam.	Body Drill	Tap Drill
0–80*	.060	52	56
1–64†	.073	49	53
1–72*	.073	49	53
2–48	.086	44	51
2–56†	.086	44	51
2–64*	.086	44	50
3–40	.099	39	47
3–48†	.099	39	47
3–56*	.099	39	46
4–32	.112	33	45
4–36	.112	33	44
4–40†	.112	33	44
4–48*	.112	33	42
5–30	.125	30	40
5–32	.125	30	40
5–36	.125	30	39
5–40†	.125	30	39
5–44*	.125	30	38
6–30	.138	28	36
6–32†	.138	28	36
6–36	.138	28	33
6–40*	.138	28	33
7–28	.150	24	32
7–30	.150	24	31
7–32	.150	24	30
7–36	.150	24	29
8–24	.164	19	30
8–30	.164	19	30
8–32†	.164	19	29
8–36*	.164	19	29
9–24	.176	16	29
9–28	.176	16	28
9–30	.176	16	27
9–32	.176	16	25
10–24†	.190	11	26
10–30	.190	11	22
10–32*	.190	11	21
11–24	.203	6	21
11–28	.203	6	17

Tap	Diam.	Body Drill	Tap Drill
11–30	.203	6	17
12–20	.216	2	19
12–22	.216	2	17
12–24†	.216	2	17
12–28*	.216	2	15
13–20	.229	A	15
13–22	.229	A	15
13–24	.229	A	13
14–20	.242	D	13
14–22	.242	D	11
14–24	.242	D	9
15–18	.255	F	10
15–20	.255	F	8
15–22	.255	F	6
15–24	.255	F	5
16–16	.268	I	7
16–18	.268	I	6
16–20	.268	I	5
16–22	.268	I	3
17–16	.282	L	6
17–18	.282	L	2
17–20	.282	L	2
18–16	.295	N	2
18–18	.295	N	1
18–20	.295	N	B
20–18	.321	P	F
20–20	.321	P	G
22–16	.347	S	I
22–18	.347	S	K
24–16	.374	V	M
24–18	.374	V	N
26–14	.400	Y	O
26–16	.400	Y	P
28–14	.426	7/16	R
28–16	.426	7/16	S
30–14	.453	29/64	U
30–16	.453	29/64	V

*A table of Metric Screw Threads will be found in the Starrett Book, Vol. II, page 136.

TAP DRILLS

Pipe Threads and Specifications

Brass, iron and steel pipe are made to the nominal sizes shown in the left column. The threads used in this country are known as Briggs standard, V-form, 60 deg. with a taper of ¾ in. per foot. The actual diameter of the pipe is in all cases larger than the nominal diameter.

Size	Threads per Inch	Tap Drill	Outside Diameter	Inside Diameter	Size	Threads per Inch	Tap Drill	Outside Diameter	Inside Diameter
1/8	27	21/64	3/32	17/64	3	8	3 3/16	3½	3 1/16
¼	18	17/64	35/64	23/64	3½	8	3 11/16	4	3 35/64
3/8	18	9/16	11/16	½					
½	14	11/16	27/32	5/8	4	8	4 3/16	4½	4 1/64
¾	14	29/32	1 1/16	53/64	4½	8	4 11/16	5	4½
1	11½	1 1/8	1 21/64	1 3/64	5	8	5¼	5 9/16	5 3/64
1¼	11½	1 15/32	1 43/64	1 25/64	6	8	6 5/16	6 5/8	6 1/16
1½	11½	1 23/32	1 29/32	1 5/8	7	8	—	7 5/8	7 1/32
2	11½	2 3/16	2 3/8	2 5/64	8	8	—	8 5/8	7 63/64
2½	8	2 9/16	2 7/8	2 15/32	9	8	—	9 5/8	8 61/64
					10	8	—	10¾	10 1/64

STOVE BOLT THREADS

The threads are U. S. Std. form, rolled and a loose fit.

Diameter	Threads per Inch	Wrench Opening For Nut	Diameter	Threads per Inch	Wrench Opening For Nut
1/8	32	5/16	¼	18	½
5/32	28	11/32	5/16	18	9/16
3/16	24	3/8	3/8	16	5/8
7/32	22	7/16			

WOOD SCREW SPECIFICATIONS

Wood screws and machine screws are made to the same gage and corresponding numbers will be the same size. The body drill listed is the nearest size over the outside diameter of the shank of the screw. For more clearance use one or two sizes larger. The small drill size is recommended practice. Unusual length screws or unusual woods may require a deviation from the sizes listed. Countersinks for flat head screws should be 82 degrees.

Number	Diameter	Threads per Inch	Body Drill	Small Drill	
				Hard Wood	Soft Wood
0	.058	32	52	56	63
1	.071	28	49	55	56
2	.084	26	44	52	55
3	.097	24	40	49	52
4	.110	22	35	45	51
5	.124	20	30	42	48
6	.137	18	28	38	45
7	.150	16	24	33	42
8	.163	15	19	31	40
9	.176	14	16	30	35
10	.189	13	12	28	32
11	.203	12	6	25	31
12	.216	11	2	20	30
13	.229	11	15/64	16	28
14	.242	10	1/4	14	26
15	.255	10	17/64	12	23
16	.268	9	9/32	8	19
17	.282	9	19/64	4	17
18	.295	8	19/64	3	16
19	.308	8	5/16	1	12
20	.321	8	21/64	15/64	8
21	.334	8	11/32	1/4	6
22	.347	7	23/64	17/64	2
23	.361	7	3/8	17/64	15/64
24	.374	7	3/8	17/64	15/64
25	.387	7	25/64	9/32	15/64
26	.400	6	13/32	19/64	1/4
27	.413	6	27/64	19/64	1/4
28	.426	6	7/16	5/16	17/64
29	.439	6	29/64	21/64	17/64
30	.453	6	29/64	11/32	9/32

Courtesy of *Motor World*.

TAPS

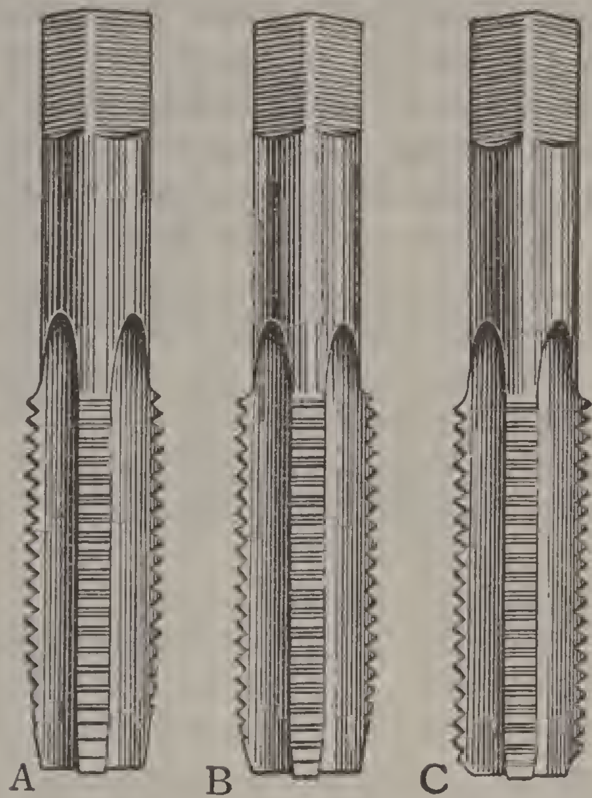


FIG. 74—STARTING, PLUG AND BOTTOMING TAPS

The commonest form of hand tap is the plug tap, shown in Fig. 74B. This tap has a bevel or chamfer at its outer end, the bevel extending back about 5 to 6 threads. This taper enables the tap to enter the hole easily, each succeeding tooth deepening the cut of thread until the tool has entered to a point above the taper, where it cuts the full size of the tap. When, however, it is desired to tap to the bottom of a blind hole, that is, one which does not go all the way through—as for cap or stud bolts—a bottoming tap, shown in Fig. 74C, is used. Usually a blind hole is first tapped as deep as possible with a plug tap and the job finished with a bottoming tap. On large holes use is often made of a starting tap, Fig. 74A, which cuts part of the thread; the plug tap is used next and, if necessary, a bottoming tap is employed to finish the job.

USING TAPS

Hand taps can be used either in a tap wrench, in tapping attachments, or a chuck for machine tool work. When using it as a hand tool it should be placed in a tap wrench, rather than in a monkey wrench, as the latter exerts pressure only on one side with the result that a clean, true thread is difficult to obtain. In addition to the difficulty encountered in obtaining a true thread when a monkey wrench is used on a tap, even more serious trouble often results from the breaking of the tap. When starting a tap, a small try-square can often be used to advantage to make sure that the tap is starting square with the surface of the work. The square should also be used occasionally during the work to check squareness.

When tapping, it is customary to turn the tap to the right (or left, as the case may be), cut a little way and then back off or reverse a turn or so in order to give the chips a chance to fall away from the tap and work out through the flutes of the trap. Lubricant—as was recommended for drilling—should be used freely.

REGRINDING TAPS

When taps are reground, care should be taken to see that the same amount of metal is ground from each flute. For grinding the cutting edge of a tap, the edge or corner of the grinding wheel should be dressed off to the approximate radius of the flute so that the hook, or under-cut, is preserved. Grind one edge of the tap at a time, removing the same amount from each flute. The tap should be passed under and the cutting edge brought against the wheel with an easy, light pressure. Be very careful not to grind the heel or rear of the next land. Care should also be taken not to draw the temper of the tool. The chamfer of approximately five threads on the nose of a plug tap should be carefully preserved. Figure 75 shows an end view of a tap and gives the names of the various parts.

Note that the lands, shown in Fig. 75, are all relieved or backed off. This is to make the tap cut easier by reducing friction in the hole and at the same time providing an actual cutting edge to form the thread. Land X in Fig. 75 shows what is called "ordinary relief", and the dotted line on land Y, what is known as straight angle relief. Some taps are made with no relief at all on the lands. Taps of this

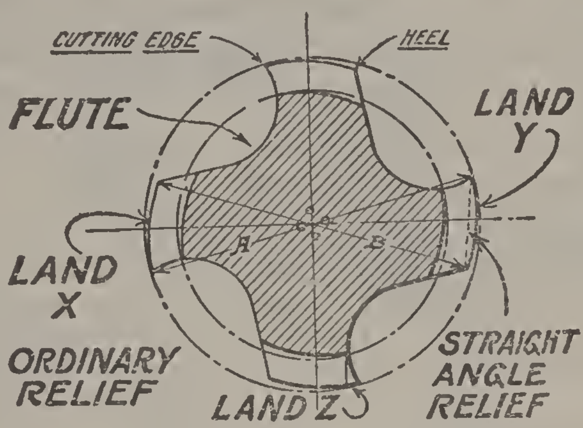


Fig. 75—Section through Tap

type will maintain their size longer and will permit regrinding as long as the land is heavy enough to stand the cut. They are, however, harder to use as the cutting edges tend to bind in the hole. Taps intended to be used in brass and similar soft metals have a rake to the cutting edge instead of a hook. That is, instead of being under-cut, the angle of the cutting edge is in an opposite direction as shown by land Z in Fig. 75. For tapping holes in copper and similar stringy metals, taps with staggered teeth are obtainable. A tap of this sort may be made by cutting out alternate teeth on each flute, thus providing additional space and freedom for chip clearance.

To Remove a Broken Tap

Motor machinists and repair men are particularly liable to meet with trouble from taps breaking. While there are devices made specially for removing the broken portion of a tap from a hole, they are not always available or satisfactory.

In such cases the broken tap may be removed by working as indicated in Fig. 76.

Two men are needed to remove a broken tap in this manner. Using blunt chisels, or drifts, which will not break the lands, they drive simultaneously and on opposite lands. Clear the hole of chips and use light blows at the start. Nitric acid may also be used for loosening a tap, diluting it to the proportion of one part of acid to two parts of water and pouring it into the hole. When this method is followed, the hole should be thoroughly washed out with water, or better with household ammonia, to prevent the acid continuing to eat into the threads.

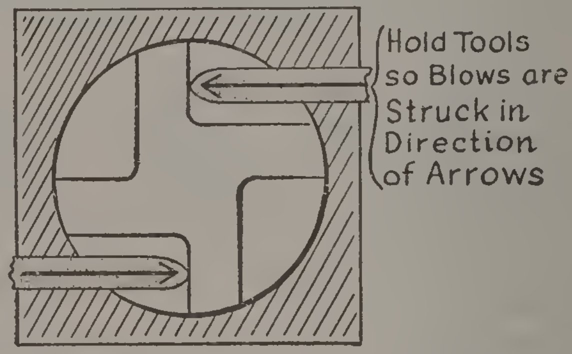


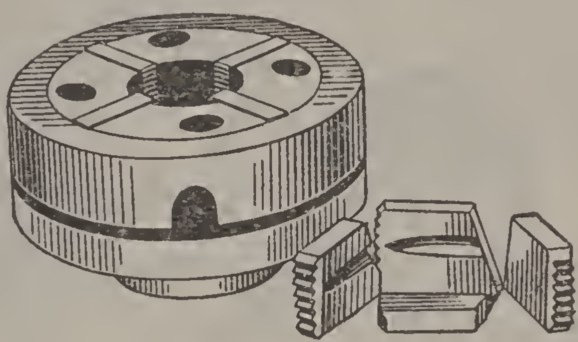
Fig. 76—Proper Method of Removing Broken Tap by Working Simultaneously on Both Sides

USE AND CARE OF DIES

The function or work of a threading die or screw plate is exactly the reverse of that performed by a tap. Whereas a tap cuts an internal thread, as for example on a nut, the corresponding die is used to cut the proper thread on a bolt to fit that nut.

The following are the common forms of dies: Die stock die, solid die, spring threading die, adjustable round-split die, rethreading die, and self-opening dies of various designs. The die stock die is made with inserted chasers—or cutters—which can be adjusted to size and can be resharpened by grinding on the cutting edge. Four chasers are generally used in this type of die, though some styles have two chasers

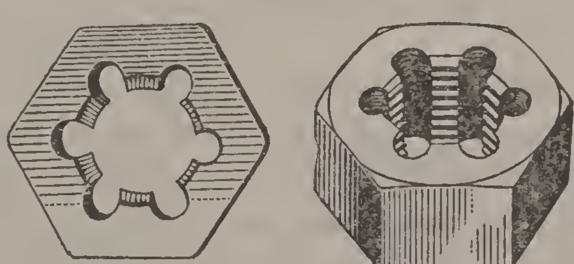
with four cutting edges. The chasers are held in place by screws on the face, set in on an angle and bearing in a slot at the sides. Two sets of screws in the rear of the plate provide means of adjusting the die to size, the screws moving the ring which holds the chasers in place and bears on the taper at the back of each chaser, causing them to move in or out as the work may require.



Die Stock Die and Extra Chasers

For sharpening, the chasers are removed from the die and are ground on the cutting edge, care being taken to remove the same amount of metal from each chaser. If, as is seldom the case, the relieved face—or side opposite the cutting edge—has to be touched up, great care must be taken to see regular angle of relief is maintained as in tap-grinding.

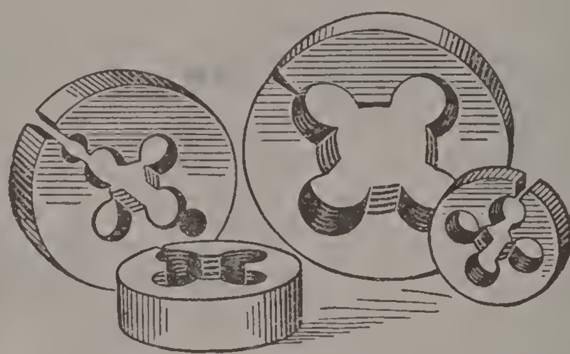
The solid die has no provision for adjustment and can be ground only on the chamfer or bevel at the front of the die. They are used exclusively for renewing worn threads and may be held in special holder or—as they are very often made hexagonal—in a common wrench.



Solid and Rethreading Die

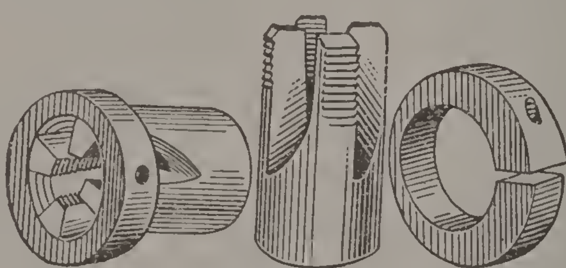
In the adjustable round-split die, compensation for wear is obtained by means of screws in the die stock which close in the die. These dies are sharpened in a manner similar to that used with solid and rethreading dies.

A spring threading die is adjusted by springing in the cutting edges by means of a threaded or plain collar which moves back on a taper. Resharpening is done by grinding



Adjustable Round-Split Die

in the chamfer and also by grinding the cutting edges with a small, thin wheel which will go down into the slots. Spring threading dies may be held either in a chuck or tap wrench or in a special holder.

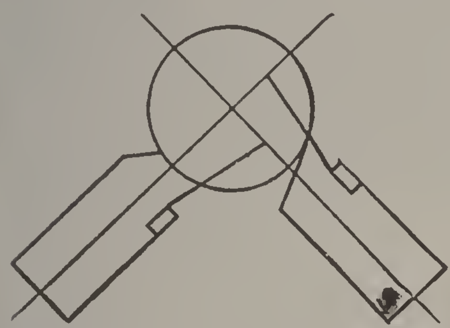


Spring Threading Dies

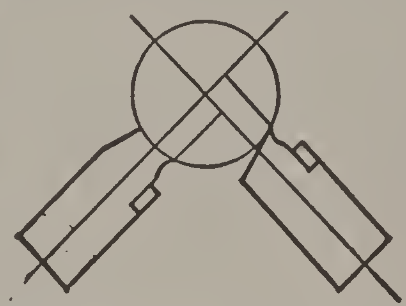
Chasers for dies are either milled or tapped. Milled chasers—those having teeth formed by a hob—are ground on the throat or chamfer and the cutting edge—where the type of die permits—is occasionally ground to keep it clean and sharp. Tapped chasers are ground on the cutting edges instead of on the throat or chamfer.

In sharpening dies the following points are worth remembering: Grind all cutting points uniformly; keep the angle for relief, chamfer, etc., as near the original as possible.

For cast iron, machine and ordinary cold rolled steel and for malleable iron, ordinary straight grinding of the chasers



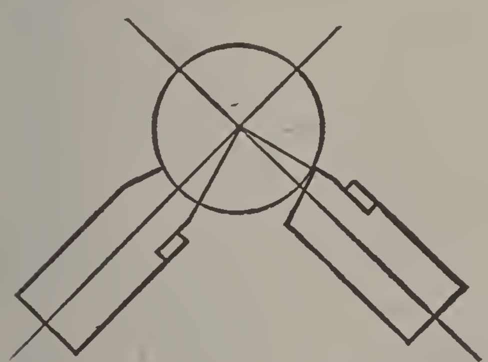
Ordinary Straight Ground Chasers



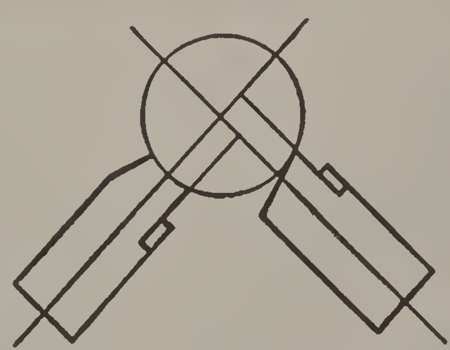
Chasers Ground with Radial Hook

is satisfactory. For tough materials, such as copper, aluminum, nickel, steel, etc., grind the chasers with a slight radial hook. Just how much hook to use can be determined only by experiment or experience.

For threading cast or bar brass, rubber or fiber, the chasers should be ground toward the center on an upward angle. For bronze, tool steel, steel and brass tubing, etc., the chasers should be ground so as to produce an angular hook.



Chasers Ground toward Center



Chasers Ground with Angular Hook

SCREW THREADS

Screw thread standardization was first attempted through the efforts of Sir Joseph Whitworth in 1841 and of William Sellers in 1864. Whitworth's work resulted in the Whitworth thread standard used in England and Sellers' work culminated in what was first known as the Franklin Institute standard and later the United States Standard thread. The United States Standard which has long been in use in the United States provides for a V-shaped thread with angles of 60 degrees and with flat bottoms and tops equal to one-eighth the height of the triangle. This took the place of the old V-thread with sharp threads which were not as strong. The sharp thread has been obsolete for a good many years. The United States Standard also provided for a definite number of threads to the inch for each diameter.

With the coming of the automobile, it was found desirable to use threads with a finer pitch than called for by the United States Standard and in the early days the Association of Licensed Automobile Manufacturers evolved a standard of finer pitched threads known as the A. L. A. M. standard or sometimes the automobile standard. Later this standard was placed in the custody of the Society of Automotive Engineers and it has since been known as the S. A. E. thread standard.

Both the United States Standard and the S. A. E. standard start with ¼ in. and there is no provision for smaller diameters. Therefore, for small work machine screw threads are used. The American Society of Mechanical Engineers some years ago evolved a machine screw standard which was known as the A. S. M. E. standard and it provided for a definite number of threads per inch for each size and also standardized the gage numbers of the screws. But the standard set was too fine for general use and the standard was not generally adhered to with the result that there has always been a good deal of confusion and multiplicity of threads in machine screw sizes. Another bad feature was that machine screw

sizes ran up over 1/4 in. and overlapped the other sizes to a considerable extent. For instance a No. 14 screw is almost the same size as a 1/4 in. screw.

In other industrial fields, other screw standards have been worked out and unfortunately these standards were not always alike. The result of all this was that when the United States entered the world war and all industries had to unite to turn out munitions, machinery and equipment, there was a great deal of confusion, a lack of real standardization and no authority for tolerances or even pitches. Accordingly, in 1918, Congress provided for the formation of a National Screw Thread Commission which would cover the ground in all the industries and which would set up standards which could be adhered to in every shop in the country.

The work of this commission, in 1924, is not yet completed, but a great mass of duplication and confusion has already been removed. While the findings of the Commission so far have neither been adopted by Congress nor by any engineering body, nevertheless, many manufacturers, tool makers and designers are following these findings.

The greatest accomplishment of the Commission is the establishment of a National Fine Thread Series and a National Coarse Thread Series. The Fine Thread Series is the same number of threads per inch as the S. A. E. standard from 1/4 in. upward, and the inclusion in the same series of certain of the old machine screw sizes up to No. 12. The National Coarse Thread Series is the same as the United States Standard with the addition of machine screw sizes below 1/4 in., the threads being coarser than in the other series. The S. A. E. Standard still includes an extra fine thread for airplane work which has not been included in the standards of the Commission.

In addition to the standardization of threads, the Commission has standardized on tolerances, gages, etc. Already drill makers are marketing drills made the exactly correct size for certain taps and more of these will appear on the

market as the standards become better known. In the National Coarse and Fine Thread Tables (See pages 97, 98), these special size drills will be found marked with an asterisk (*). Several great automobile companies, notably the General Motors Company, have adopted the findings of the Commission as standard.

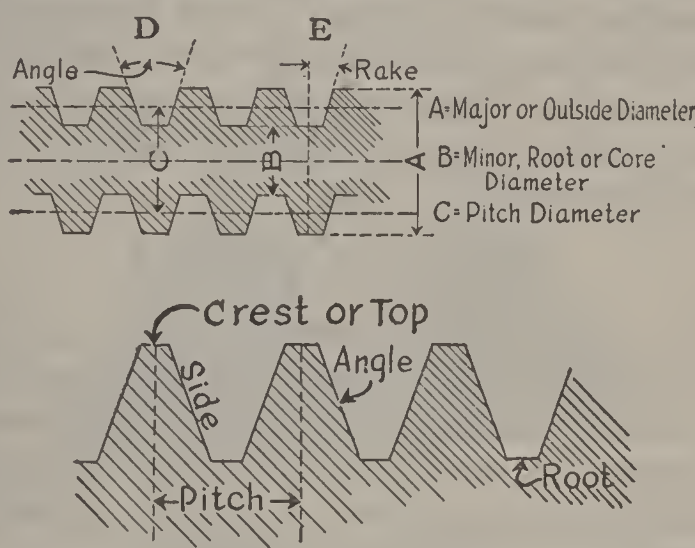
In addition to the National Series Threads, Coarse and Fine, there are two other screw standards used on the automobile. One of these is the standard pipe thread, used for gasoline and air connections, spark plugs, casting plugs, drain plugs, etc., and the stove bolt threads. Both of these standards are listed in this volume.

The screw threads commonly used in motor construction are as follows: The "V" thread, U. S. Standard, S. A. E. (Society of Automotive Engineers) Standard, Whitworth or British Standard, A. S. M. E. (American Society of Mechanical Engineers) Standard, Square, Acme, Worm, B. S. F. (British Standard Fine) Standard, B. A. S. (British Association Standard) for very small screws, and the Metric. In addition, pipes and tubing are cut with American (Briggs' "Standard") Pipe Thread and British Standard Pipe Threads. The S. A. E. screw thread, formerly known as the A. L. A. M. thread, and the A. S. M. E. Standard are the same form of thread as the U. S. Standard, though what is commonly called the pitch, or number of threads to the inch, of the S. A. E. thread is much higher than that of the U. S. Standard thread. To these must now be added the new "National Series Threads" which will probably ultimately render obsolete all of the old threads.

Figures 77 and 78 show the principal parts of a screw and thread and will aid in making clear the differences in the different types of threads in use.

The angle of the thread (D) is the angle the sides of the thread make with each other, as shown in Fig. 77. The rake (E) is the angle between the side of thread and a line at right angles to the axis of the screw. The major or out-

side diameter (A) is the overall dimension, measuring to the outside of the threads. The minor, core or—as it is usually called—the root diameter (B) is the diameter of the screw at the base of the threads and is equivalent to the major diameter minus twice the depth of the thread. The pitch diameter (C) is the major diameter minus a single depth of thread. Pitch, see Fig. 78, is really the distance between two adjacent threads measured on a line parallel to the axis of the screw, though the term is often incorrectly



Figs. 77–78—Principal Parts of a Screw—Principal Parts of a Thread

used to indicate the number of threads to the inch. The proper name for "pitch" as meaning the number of threads to the inch, or the distance a nut will travel on a screw when the screw is given one turn, is lead. In a single thread screw, pitch and lead are equal, because the nut will move from one thread to the next with each turn of the screw, but in a double or triple thread screw the nut will move over two or three threads for each turn of the screw and in such screws the lead is two or three times the pitch.

Double and triple thread screws*, often called multiple thread screws, are not commonly used in automotive prac-

*See page 141, Vol. II, Starrett Books.

tice. They may be used to advantage, however, on parts having thin walls or small diameters and which therefore will not stand a deep or coarse thread with a high lead. On such a part, where it is desirable for the nut or screw to move rapidly, a multiple thread will often solve the difficulty, the number of threads to the inch being greatly increased above normal, but are cut to only half the usual depth. For example, if a single thread with a ¼ in. lead and but a half or ⅛ in. pitch depth is cut on a small bolt and then another similar thread is cut between the threads already cut, the nut will move forward the full ¼ in. lead for each turn of the bolt, but will move over two threads in doing so. Such a thread would be called a "¼ in. lead by ⅛ in. pitch, double".

The "V" Thread

In the "V" thread the top or crest and the root are both theoretically sharp, but in actual practice there is formed a little flat of about 1/25 inch. The angle is 60° and the depth (d) is .866 times the pitch (p), the pitch being obtained by dividing 1.000 by the number of threads to the inch. The formula for the "V" thread is as follows:

$$p = \text{pitch} = \frac{1}{\text{No. Threads per Inch}}$$

$$d = \text{depth} = p \times .86603.$$

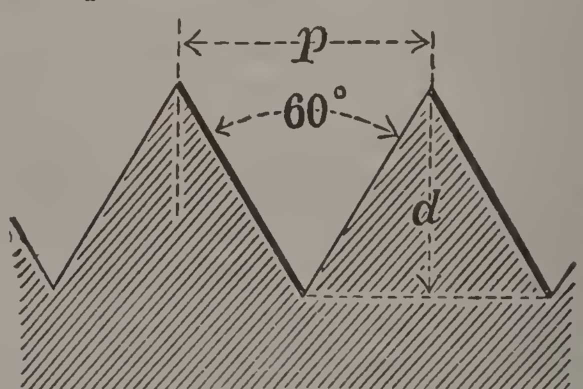


Fig. 79—Section of a "V" Thread

U. S. Standard Thread*

The U. S. Standard is probably the thread most commonly used in machine shop practice and is an outgrowth of the "V" thread which has now been rendered almost obsolete, though it is still used as a pipe thread where it is known as the Briggs' Standard. The disadvantages of the "V" thread were that it was an impossible thread to reproduce economically because with the slightest wear on the thread cutting

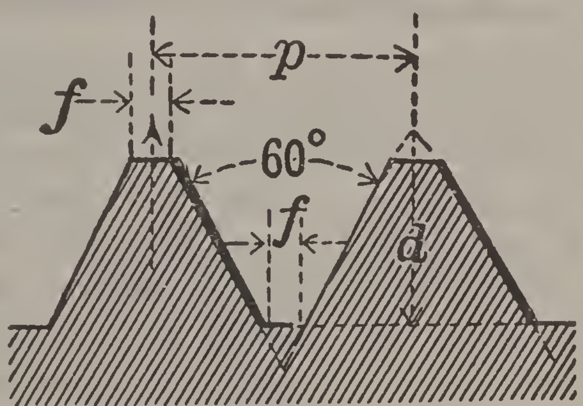


Fig. 80—Section of U. S. Standard Thread

tools the crest and the root of the thread could not be made sharp. Consequently, when a "V" threaded nut and screw were assembled the bearing surfaces were only at the crest and root—instead of the sides—and a great tendency to unscrew was thereby developed. In the U. S. thread, where there is a flat at both the crest and the root of the thread, the sides become the bearing surface and the thread is not only much more economically produced, but the tendency to unscrew is overcome.

The angle of the U. S. Standard thread is 60° as in the "V" thread. The width of the flats (f) is ⅛ the pitch. The depth (d) is .6495 times the pitch. The formula for the U. S. Standard thread is as follows:

*See pages 137–138, Vol. II, Starrett Books.

$$p = \text{pitch} = \frac{1}{\text{No. Threads per Inch}}$$

$$n = \text{number of threads per inch}$$

$$d = \text{depth} = p \times .6495 \text{ or } \frac{.6495}{n}$$

$$f = \text{flat} = \frac{p}{8}$$

S. A. E. Standard Thread*

The S. A. E. Standard thread is identical with what was known as the A. L. A. M. thread. In this standard the U. S. form of thread is used, the only difference being in the threads per inch for certain diameters. The finer pitch of the S. A. E. thread makes it particularly well adapted to automobile practice as it is less likely to work loose from vibration than is the comparatively coarse thread of the U. S. Standard. The formula of the S. A. E. thread is the same as for the U. S. Standard.

Metric Threads†

The metric thread standard is based on the U. S. Standard, but to provide clearance at the root of the thread a radius not exceeding 1/16 of the height of the thread is recommended. There are really two metric standards, the French and what is called the International. The formula in each case is the same as for the U. S. Standard.

Whitworth Thread Standard‡

The Whitworth thread, although used to some extent in locomotive practice in this country, is peculiar to English cars and machine parts of British origin.

The angle in the Whitworth thread form is 55°, as compared with 60° in the U. S. Standard, and the crest and root

*See page 134, Vol. II, Starrett Books. †See page 136, Vol. II, Starrett Books.
‡See page 139, Vol. II, Starrett Books.

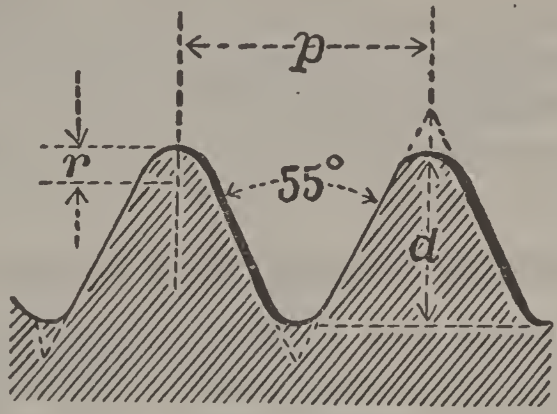


Fig. 80A—Section of a Whitworth Thread

are rounded, rather than flattened, to a radius equal to .1373 times the pitch. The formula is as follows:

$$p = \text{pitch} = \frac{1}{\text{No. of Threads per Inch}}$$

$$\mathbf{d} = \text{depth} = p \times .64033$$

$$\mathbf{r} = \text{radius} = p \times .1373$$

Square Thread Standard*

The square thread is so made that the width between the threads and the depth of the thread are the same, and are

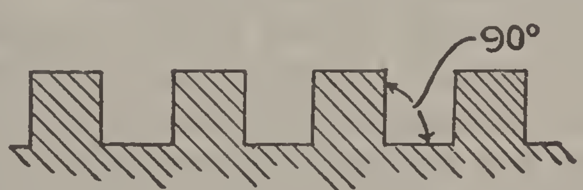


Fig. 81—Section of a Square Thread

equal to ½ the pitch. There is no recognized standard for this thread and it is being rapidly superseded by the Acme thread which is easier to cut. While theoretically, a square

*See page 135, Vol. II, Starrett Books.

thread, as its name implies, and as is shown in Fig. 81, has straight sided threads, in actual practice it is customary to leave a slight amount of rake—2° to 5°—so the sides of the thread are not exactly parallel, or at right angles to the axis of the screws.

Acme Standard Thread*

The Acme thread was adapted from the worm thread, and was introduced to overcome the disadvantages and difficulties in cutting the square thread. It is a little shallower

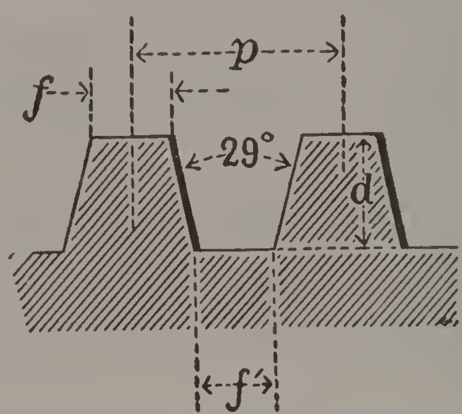


Fig. 82—Section of Acme Thread

than the worm thread, but the same depth as the square thread and much stronger than the latter.

The angle of the Acme thread is 29°, as shown in Fig. 82. The depth of the thread is ½ the pitch plus .010 in. for clearance. The root diameter of the screw is the nominal diameter of the screw less the pitch plus .020 in., the nut for this thread being made .020 in. oversize. The formulae for the Acme standard thread are as follows:

Width of Point of Tool for Screw or Tap Thread

$$= \frac{.3707}{\text{No. of Threads per Inch}} - .0052$$

*See page 136, Vol. II, Starrett Books.

Width of Screw or Nut Thread

$$= \frac{.3707}{\text{No. of Threads per Inch}}$$

Diameter of Tap = Diameter of Screw $+ .020$

Root Diameter of Tap or Screw

$$= \text{Diameter of Screw} - \left(\frac{1}{\text{No. of Threads per Inch}} + .020\right)$$
$$= p + .020$$

Depth of Thread

$$= \frac{1}{2 \times \text{No. of Threads per Inch}} + .010 = \tfrac{1}{2}\,p + .010$$

The Acme standard has no standard number of threads per inch for certain diameters as have all other standards and should, therefore, never be specified as a "3/8 or 1 in., etc., Acme". To identify an Acme screw the pitch diameter or the number of threads per inch should be given.

TAP DRILL SIZES

75% DEPTH OF THREAD

A bolt inserted in an ordinary nut which has only one-half of a full depth of thread, will break before stripping the thread. Also, a full depth of thread, while very difficult to obtain, is only about 5% stronger than a 75% depth.

These tables give the exact size of the hole, expressed in decimals, that will produce a 75% depth of thread and also the nearest regular stock drill to this size. Holes produced by these drills are close enough for any commercial tapping.

$$\text{Diam. of Tap} - \frac{.974}{\text{No. Threads per Inch}} = \text{Diam. of Hole}$$

Tap Drill Sizes—75 Per Cent Depth Thread
Machine Screw Threads

Tap Size	Threads per In.	Diam. Hole	Drill	Tap Size	Threads per In.	Diam. Hole	Drill	Tap Size	Threads per In.	Diam. Hole	Drill
0	*80	.048	3/64	7	*36	.124	1/8	16	*22	.224	2
1	*72	.060	53	7	32	.121	31	16	20	.219	7/32
1	64	.058	53	7	30	.119	31	16	18	.214	3
2	*64	.071	50	8	*36	.137	29	18	*20	.245	D
2	56	.069	51	8	32	.134	29	18	18	.240	B
3	*56	.082	46	8	30	.132	30	20	*20	.271	I
3	48	.079	47	9	*32	.147	26	20	18	.266	17/64
4	*48	.092	42	9	30	.145	27	22	*18	.292	L
4	40	.088	44	9	24	.136	29	22	16	.285	9/32
4	36	.085	44	10	32	.160	21	24	18	.318	O
5	*44	.103	38	10	*30	.158	22	24	*16	.311	5/16
5	40	.101	39	10	24	.149	26	26	*16	.337	R
5	36	.098	40	12	*28	.181	15	26	14	.328	21/64
6	*40	.114	33	12	24	.175	17	28	16	.363	23/64
6	36	.111	34	14	*24	.201	7	28	*14	.354	T
6	32	.108	36	14	20	.193	10	30	16	.389	25/64
								30	*14	.380	V

*A. S. M. E. Standard.

Tap Drill Sizes—75 Per Cent Depth Thread
Special and Standard Threads

Tap Size	Threads per In.	Diam. Hole	Drill	Tap Size	Threads per In.	Diam. Hole	Drill	Tap Size	Threads per In.	Diam. Hole	Drill
1/16	72	.049	3/64	1/4	32	.220	7/32	7/8	*14	.805	13/16
1/16	64	.047	3/64	1/4	*28	.215	3	7/8	12	.794	51/64
1/16	60	.046	56	1/4	27	.214	3	†7/8	9	.767	49/64
5/64	72	.065	52	1/4	24	.209	4	15/16	12	.856	55/64
5/64	64	.063	1/16	†1/4	20	.201	8	15/16	9	.829	53/64
5/64	60	.062	1/16	5/16	32	.282	9/32	1	27	.964	31/32
5/64	56	.061	53	5/16	27	.276	J	1	*14	.930	15/16
3/32	60	.077	5/64	5/16	*24	.272	9/32	1	12	.919	59/64
3/32	56	.076	48	5/16	20	.264	17/64	†1	8	.878	7/8
3/32	50	.074	49	†5/16	18	.258	F	1 1/16	8	.941	15/16
3/32	48	.073	49	3/8	27	.339	R	1 1/8	*12	1.044	1 3/64
7/64	56	.092	42	3/8	*24	.334	Q	†1 1/8	7	.986	63/64
7/64	50	.090	43	3/8	20	.326	21/64	1 3/16	7	1.048	1 3/64
7/64	48	.089	43	†3/8	16	.314	5/16	1 1/4	*12	1.169	1 11/64
1/8	48	.105	36	7/16	27	.401	Y	†1 1/4	7	1.111	1 7/64
1/8	40	.101	38	7/16	24	.397	X	1 5/16	7	1.173	1 11/64
1/3	36	.098	40	7/16	*20	.389	25/64	1 3/8	12	1.294	1 19/64
1/8	32	.095	3/32	†7/16	14	.368	U	1 3/8	6	1.213	1 7/32
9/64	40	.116	32	1/2	27	.464	15/32	1 1/2	*12	1.419	1 27/64
9/64	36	.114	33	1/2	24	.460	29/64	†1 1/2	6	1.338	1 11/32
9/64	32	.110	35	1/2	*20	.451	29/64	1 5/8	5 1/2	1.448	1 29/64
3/32	40	.132	30	†1/2	13	.425	27/64	†1 3/4	5	1.555	1 35/64
5/32	36	.129	30	1/2	12	.419	27/64	1 7/8	5	1.680	1 11/16
5/32	32	.126	1/8	9/16	27	.526	17/32	†2	4 1/2	1.783	1 25/32
11/64	36	.145	27	9/16	*18	.508	33/64	2 1/8	4 1/2	1.909	1 29/32
11/64	32	.141	9/64	†9/16	12	.481	31/64	2 1/4	4 1/2	2.034	2 1/32
3/16	36	.161	20	5/8	27	.589	19/32	2 3/8	4	2.131	2 1/8
3/16	32	.157	22	5/8	*18	.571	37/64	†2 1/2	4	2.256	2 1/4
3/16	30	.155	23	5/8	12	.544	35/64	2 5/8	4	2.381	2 3/8
3/16	24	.147	26	†5/8	11	.536	17/32	†2 3/4	4	2.506	2 1/2
13/64	32	.173	17	11/16	16	.627	5/8	2 7/8	3 1/2	2.597	2 19/32
13/64	30	.171	11/64	11/16	11	.599	19/32	†3	3 1/2	2.722	2 3/4
13/64	24	.163	20	3/4	27	.714	23/32	3 1/8	3 1/2	2.847	2 27/32
7/32	32	.188	12	3/4	*16	.689	11/16	3 1/4	3 1/2	2.972	2 31/32
7/32	28	.184	13	3/4	12	.669	43/64	3 3/8	3 1/4	3.075	3 1/16
7/32	24	.178	16	†3/4	10	.653	21/32	3 1/2	3 1/4	3.200	3 3/16
15/64	32	.204	6	13/16	12	.731	47/64	3 5/8	3 1/4	3.325	3 5/16
15/64	28	.200	8	13/16	10	.715	23/32	3 3/4	3	3.425	3 7/16
15/64	24	.194	10	7/8	27	.839	27/32	3 7/8	3	3.550	3 9/16
				7/8	18	.821	53/64	4	3	3.675	3 11/16

*S. A. E. Standard. †U. S. Standard.

UNITED STATES STANDARD SCREW THREADS
(Sizes above 3 inches are U. S. S. Form)

Diameter	Threads Per Inch	Diameter at Root of Thread	Diameter of Tap Drill	Sectional Area in Square Inches		Nuts and Bolt Heads*			
				Solid Bolt	At Root of Thread	Across Flats, Square and Hex.	Across Corners *a*		Thickness, Bolt Head
							Hexagon	Square	
1/4	20	0.1850	No. 8	0.0491	0.0269	1/2	0.5774	0.7071	1/4
5/16	18	0.2403	1/4	0.0767	0.0459	19/32	0.6856	0.8397	19/64
3/8	16	0.2938	5/16	0.1104	0.0677	11/16	0.7939	0.9723	11/32
7/16	14	0.3447	23/64	0.1503	0.0933	25/32	0.9021	1.1049	25/32
1/2	13	0.4001	27/64	0.1964	0.1257	7/8	1.0104	1.2374	7/16
9/16	12	0.4542	31/64	0.2485	0.1620	31/32	1.1186	1.3700	31/64
5/8	11	0.5069	17/32	0.3068	0.2018	1 1/16	1.2269	1.5026	17/32
3/4	10	0.6201	21/32	0.4418	0.3024	1 1/4	1.4434	1.7678	5/8
7/8	9	0.7307	49/64	0.6013	0.4230	1 7/16	1.6600	2.0329	23/32
1	8	0.8376	7/8	0.7854	0.5510	1 5/8	1.8764	2.2981	13/16
1 1/8	7	0.9394	63/64	0.9940	0.6931	1 13/16	2.0929	2.5633	29/32
1 1/4	7	1.0644	1 7/64	1.2272	0.8898	2	2.3094	2.8284	1
1 1/2	6	1.2835	1 11/32	1.7672	1.2938	2 3/8	2.7424	3.3588	1 3/16
1 3/4	5	1.4902	1 35/64	2.4053	1.7454	2 3/4	3.1754	3.8891	1 3/8
2	4 1/2	1.7113	1 49/64	3.1416	2.3000	3 1/8	3.6084	4.4194	1 9/16
2 1/4	4 1/2	1.9613	2 1/32	3.9761	3.0212	3 1/2	4.0414	4.9498	1 3/4
2 1/2	4	2.1752	2 1/4	4.9087	3.7161	3 7/8	4.4745	5.4801	1 15/16
2 3/4	4	2.4252	2 1/2	5.9396	4.6180	4 1/4	4.9075	6.0104	2 1/8
3	3 1/2	2.675	2 3/4	7.0686	5.520	4 5/8	5.3405	6.5407	2 5/16
3 1/4	3 1/2	2.8788	2 15/16	8.2958	6.5090	5	5.7735	7.0711	2 1/2
3 1/2	3 1/4	3.1003	3 11/64	9.6211	7.5492	5 3/8	6.2065	7.6014	2 11/16
3 3/4	3	3.3170	3 3/8	11.045	8.6414	5 3/4	6.6395	8.1317	2 7/8
4	3	3.5670	3 5/8	12.566	9.9930	6 1/8	7.0725	8.6621	3 1/16
4 1/4	2 7/8	3.7982	3 27/32	14.186	11.330	6 1/2	7.5055	9.1924	3 1/4
4 1/2	2 3/4	4.0276	4 3/32	15.904	12.740	6 7/8	7.9386	9.7227	3 7/16
4 3/4	2 5/8	4.2551	4 5/16	17.721	14.220	7 1/4	8.3716	10.253	3 5/8
5	2 1/2	4.4804	4 9/16	19.635	15.766	7 5/8	8.8046	10.783	3 13/16
5 1/4	2 1/2	4.7304	4 13/16	21.648	17.575	8	9.2376	11.314	4
5 1/2	2 3/8	4.9530	5 1/32	23.758	19.268	8 3/8	9.6706	11.844	4 3/16
5 3/4	2 3/8	5.2030	5 9/32	25.967	21.262	8 3/4	10.104	12.374	4 3/8
6	2 1/4	5.4226	5 1/2	28.274	23.094	9 1/8	10.537	12.905	4 9/16

*Thickness of nut equals diameter of bolt.

a. In practice these are a trifle less than shown, because the corners are slightly rounded.

NATIONAL COARSE THREAD SERIES

Diameter	Thre'ds	Major Diameter	Body Drill	Hole to Tap 83% Thread	Hole to Tap 75% Thread	Recommended Tap Drill (75–80%)	Nearest Other Tap Drill
1	64	.073	49	.0561	.0578	.0575*	53 (67%)
2	56	.086	44	.0667	.0686	.0682*	51 (82%)
3	48	.099	39	.0764	.0787	5/64	47 (76%)
4	40	.112	33	.0849	.0876	44	43 (71%)
5	40	.125	30	.0979	.1006	39	38 (72%)
6	32	.138	28	.1042	.1076	36	———
8	32	.164	19	.1302	.1336	.1324*	29 (69%)
10	24	.190	11	.1449	.1494	26	25 (75%)
12	24	.216	2	.1709	.1754	17	16 (72%)
1/4	20	.2500	1/4	.1959	.2013	8	7 (75%)
5/16	18	.3125	5/16	.2524	.2584	F	1/4 (82%)
3/8	16	.3750	3/8	.3073	.3141	5/16	———
7/16	14	.4375	7/16	.3602	.3679	U	23/64 (82%)
1/2	13	.5000	1/2	.4167	.4251	27/64	———
9/16	12	.5625	9/16	.4723	.4813	.4776*	31/64 (72%)
5/8	11	.6250	5/8	.5266	.5364	17/32	———
3/4	10	.7500	3/4	.6417	.6526	.6480*	21/32 (72%)
7/8	9	.9750	7/8	.7547	.7667	.7615*	49/64 (76%)
1	8	1.0000	1	.8647	.8782	.8723*	7/8 (77%)
1 1/8	7	1.1250	1 1/8	.9704	.9858	.9789*	63/64 (76%)
1 1/4	7	1.2500	1 1/4	1.0954	1.1108	1 7/64	———
1 1/2	6	1.5000	1 1/2	1.3196	1.3376	1 21/64	1 11/32 (72%)
1 3/4	5	1.7500	1 3/4	1.5335	1.5551	1.5453*	1 35/64 (78%)
2	4 1/2	2.0000	2	1.7594	1.7835	1 25/32	1 49/64 (81%)
2 1/4	4 1/2	2.2500	2 1/4	2.0094	2.0335	2.0225*	2 1/32 (76%)
2 1/2	4	2.5000	2 1/2	2.2294	2.2564	2 1/4	2 15/64 (82%)
2 3/4	4	2.7500	2 3/4	2.4794	2.5064	2.4939*	2 1/2 (77%)
3	4	3.0000	3	2.7294	2.7564	2.7439*	2 3/4 (77%)

*These are standard stock drill sizes and are readily obtainable.

NATIONAL FINE THREAD SERIES

Diameter	Thre'ds	Major Diameter	Body Drill	Hole to Tap 83% Thread	Hole to Tap 75% Thread	Recommended Tap Drill (75–80%)	Nearest Other Tap Drill
0	80	.060	52	.0465	.0478	3/64	56 (83%)
1	72	.073	49	.0580	.0595	53	———
2	64	.086	44	.0691	.0708	50	———
3	56	.099	39	.0797	.0816	46	45 (73%)
4	48	.112	33	.0894	.0917	.0911*	42 (68%)
5	44	.125	30	.1004	.1029	38	37 (71%)
6	40	.138	28	.1109	.1136	33	34 (83%)
8	36	.164	19	.1339	.1369	29	———
10	32	.190	11	.1562	.1596	21	22 (81%)
12	28	.216	2	.1773	.1812	15	14 (73%)
1/4	28	.2500	1/4	.2113	.2152	3	———
5/16	24	.3125	5/16	.2674	.2719	.2703*	17/64 (100%)
3/8	24	.3750	3/8	.3299	.3344	Q	21/64 (100%)
7/16	20	.4375	7/16	.3834	.3888	W	25/64 (72%)
1/2	20	.5000	1/2	.4459	.4513	.4492*	29/64 (72%)
9/16	18	.5625	9/16	.5024	.5084	.5062*	33/64 (65%)
5/8	18	.6250	5/8	.5649	.5709	.5687*	37/64 (65%)
3/4	16	.7500	3/4	.6823	.6891	11/16	———
7/8	14	.8750	7/8	.7977	.8054	.8024*	13/16 (67%)
1	14	1.0000	1	.9227	.9304	.9274*	15/16 (67%)
1 1/8	12	1.1250	1 1/8	1.0348	1.0438	1.0401*	1 3/64 (72%)
1 1/4	12	1.2500	1 1/4	1.1598	1.1688	1.1651*	1 11/64 (72%)
1 1/2	12	1.5000	1 1/2	1.4098	1.4188	1.4153*	1 27/64 (72%)
1 3/4	12	1.7500	1 3/4	1.6598	1.6688	1.6653*	1 43/64 (72%)
2	12	2.0000	2	1.9098	1.9188	1.9153*	1 59/64 (72%)
2 1/4	12	2.2500	2 1/4	2.1532	2.1687	2 5/32	———
2 1/2	12	2.5000	2 1/2	2.4032	2.4187	2 13/32	———
2 3/4	12	2.7500	2 3/4	2.6532	2.6687	2 21/32	———
3	10	3.0000	3	2.8919	2.9027	2 57/64	———

*These are standard stock drill sizes and are readily obtainable.

HOW TO USE THICKNESS GAGES

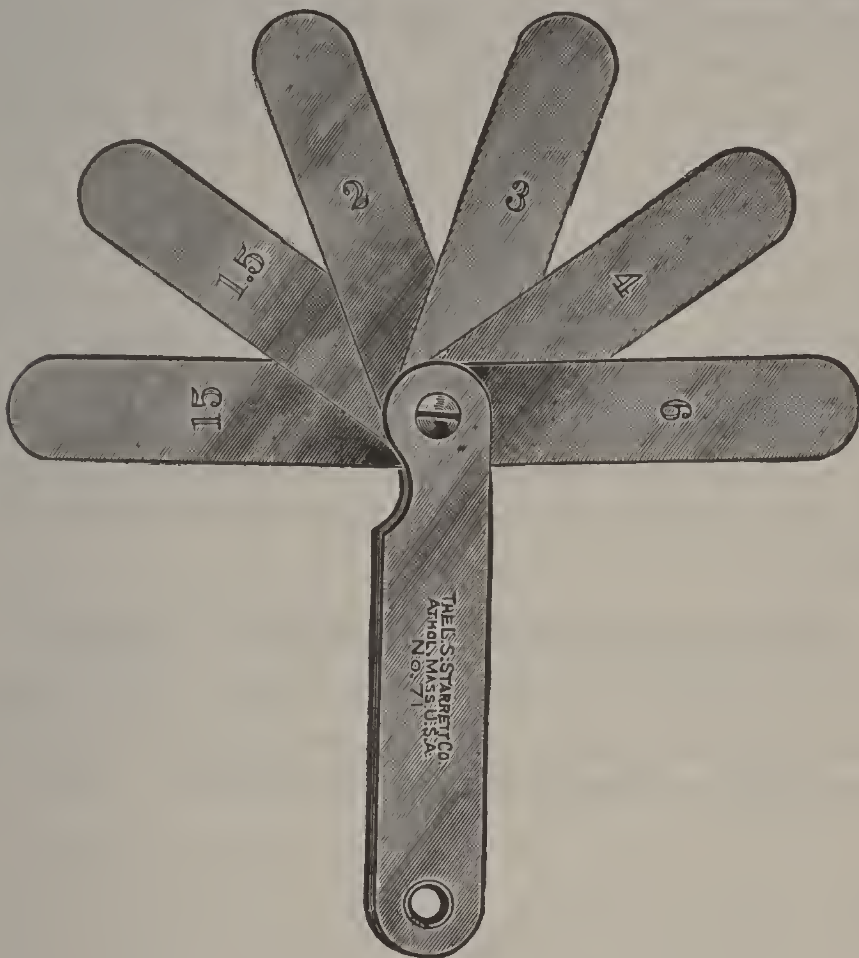


Fig. 83—Thickness Gage

One of the most useful and indispensable tools in an automobile repair shop, and at the same time one of the least expensive, is the thickness gage. See Fig. 83. This tool is of great use in making inspections of various parts of the car for wear, in eliminating guesswork and—when the necessity is imperative—it can be made to approximate the work of a number of other and much more expensive tools. Thickness gages consist of a varying number of metal leaves of different thicknesses fastened together in a convenient holder in such a manner that any combination of them may be made. The thickness of each leaf is plainly indicated and,

having found a combination of leaves that just fills the space to be measured, the sum of the thicknesses of the various leaves used indicates the width or thickness of the space. Again, it may be desired that a gap—as in a spark plug—shall be a certain amount. In such a case a leaf or leaves having the desired thickness is selected and the gap made to conform to it.

Spacing gaps in spark plugs is perhaps the commonest use of thickness gages in garage and repair shop work and is far too often a piece of guesswork. In most ignition systems, the proper width for a spark gap is 1/32 of an inch. Reference to a decimal equivalent table shows that 1/32 in. is .031 in.—a thickness easily determined on a thickness gage made for the automobile trade by placing the proper leaves together.

Setting Contact Breaker Points

Another motor—or rather ignition—part which calls for the use of a thickness gage if good work is to be done, is the contact breaker. Here the gap varies from .010 to .015,

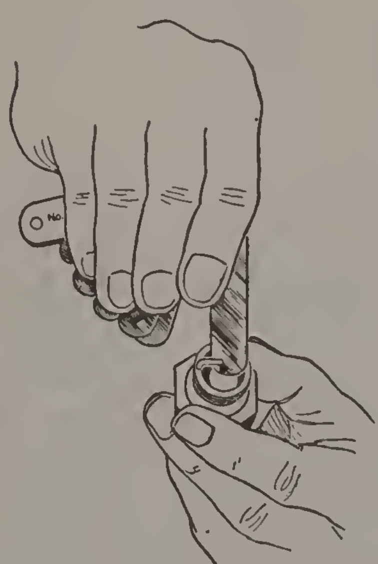


Fig. 84—Setting Spark Plug Gaps

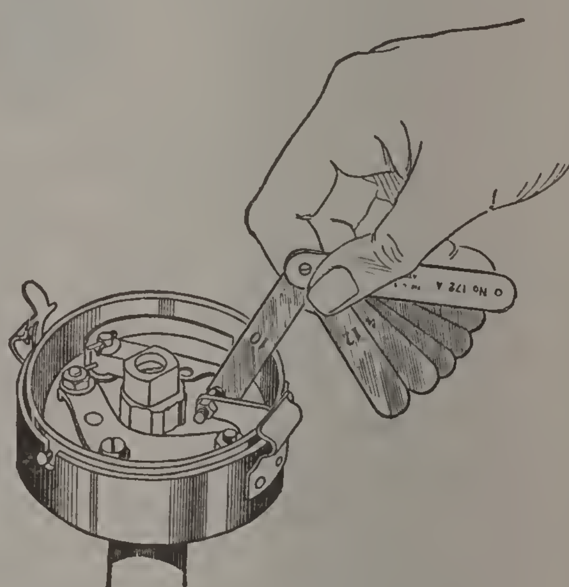


Fig. 85—Setting Breaker Contact Points

according to the type of ignition system, and the difference of a very few thousandths of an inch determines the efficiency of the motor. Never attempt to set contact breaker points without using a thickness gage.

Adjusting Tappets

Adjusting tappets or push rods is another common use for thickness gages. Tappets are usually set to give a clearance between the push rod and tappet, or between the rocker arm and the end of the valve stem, of from .002 to .005 in.—according to the make of motor—when the engine is hot. Pieces of paper are often used for this purpose, but it should be realized that paper thicknesses range from .001 to .020 inch and that such a method is little more than guesswork. Manufacturers' recommendations, as regards the amount of clearance, should be closely followed and a thickness gage invariably used.

Pistons and Piston Rings

Thickness gages may also be used, when the proper tools are not available—cylinder gage or inside micrometer—in fitting new or oversize pistons. When the work has to be done with a thickness gage, the old piston should be placed in the bore and the blades of the gage tried successively until

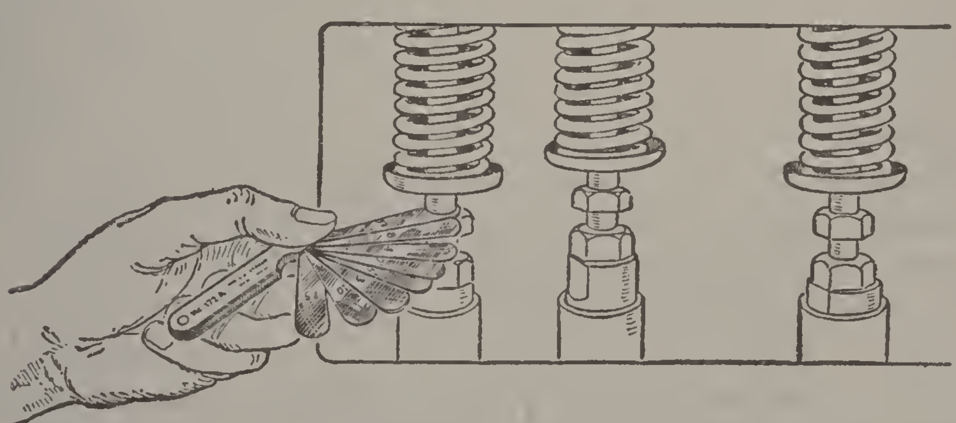


FIG. 86—ADJUSTING TAPPET OR PUSH RODS

one—or a combination—is found that will just enter between the piston and the cylinder wall. The piston is then removed, measured with an outside micrometer and the thickness of the gage blade added. The result indicates the actual diameter of the cylinder bore. A thickness gage can also be of some assistance in determining if, and how much, a cylinder is out of round, though to do a satisfactory job a cylinder gage should be employed. In using a thickness gage for this work, merely insert a new and true piston in the cylinder and test the clearance—using blades of different thicknesses if necessary—all around the circumference of the piston.

A thickness, or feeler gage as it is sometimes called, may be used to check the clearance of pistons and cylinders before assembly, even though they have been properly measured with micrometers.

Thickness gages should be used in checking the gap in both step and diagonal joint piston rings. The usual allowance for expansion of the ring by the heat of the motor is about .001 in. per inch of bore. Patent or compound rings, etc., are made with correct clearances by the manufacturer and no fitting is necessary.

Shim Fitting

Thickness gages should be used also in all work calling for fitting of shims. In replacing or scraping in connecting rod, crank or camshaft bearings, work on chassis or running gear, steering wheel spindles, and springs and shackle bolts, thickness gages can be used to determine beforehand the actual thickness of shim required. In the case of shaft bearings that have been scraped or lapped, put on the cap without any shims, draw up approximately tight and then use the thickness gage between the bearing and the shaft to find the total amount of shim required. Divide that by two to get the shim thickness for each side. Wherever shims are to be used, a thickness gage will save a great amount of trouble as well as considerable cut-and-try-work and material.

Gear Meshing

Various gears in the car are supposed to run with various amount of backlash depending on their construction, operation and work. It is possible—with the exception of extremely small gears—to mesh gears very exactly by using a feeler gage to determine the backlash.

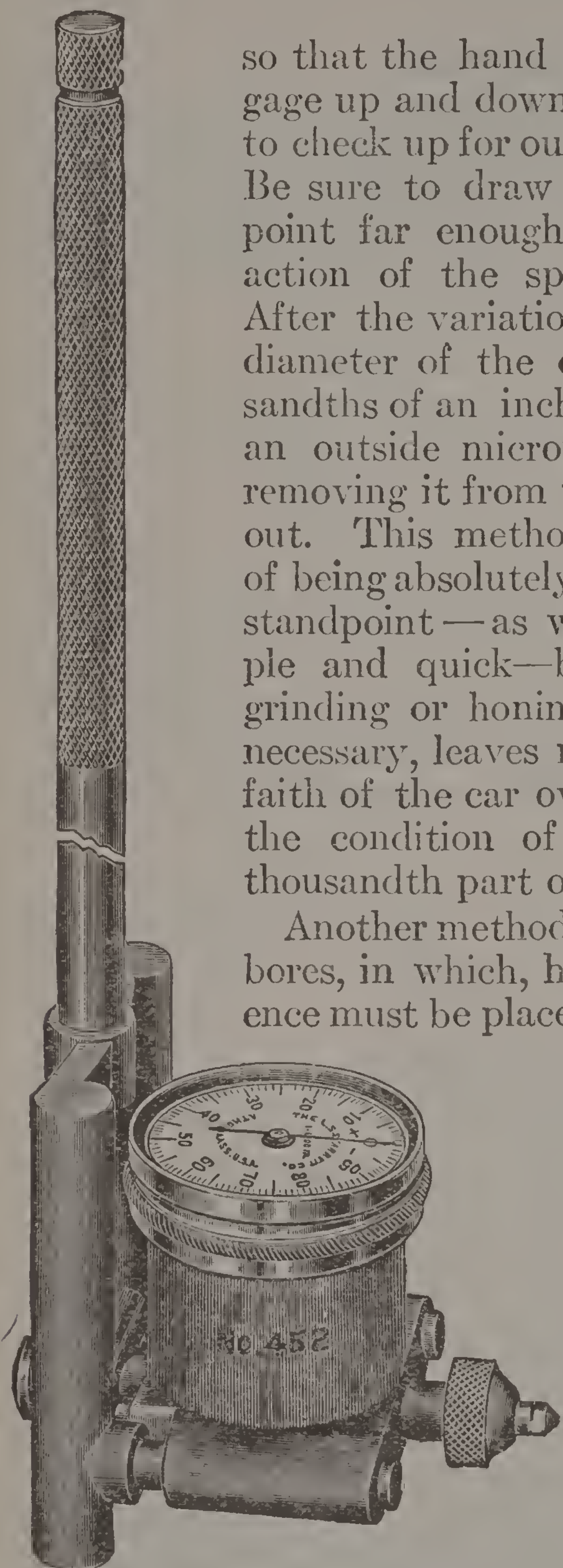
X

GAGING CYLINDER BORES

A machinist in any motor repair or service shop should be supplied with the necessary tools to enable him to test and gage cylinder bores accurately in order to determine if and how much they are tapered, out of round or scored. To do the job most efficiently and in a manner that will indicate the exact condition of the cylinder to even the most ignorant car owner, he should have a cylinder gage (Fig. 87) and outside micrometer for transferring measurements from the gage (Fig. 88).

A gage, such as is shown in Fig. 87, consists of a sled on which are two line contact points that are at all times in alignment with the walls of the cylinder. These hardened contact points record taper, eccentricity, scoring, etc., by the position or movement of the hand on the dial when the gage is moved up or down or rotated in the cylinder bore. The dial is graduated to read plus and minus by .001 in., and provision is made for gaging cylinders of any diameter between 2½ in. and 6 in. A double spring action makes the gage self-centering and absolutely non-collapsible.

The method of using the gage is indicated in Fig. 89. First place the sled against the bore of the cylinder, then set the adjustable contact point out over the edge of the bore so when the gage is set down in the cylinder the tension on the springs will show that the hand on the dial has been forced to the minus side 40 or 50 thousandths, at least enough so that both contact points are forced against the walls of the cylinder. Turn the knurled rim of the dial



so that the hand registers at 0 and move the gage up and down to test for taper, rotating it to check up for out-of-round or scored cylinders. Be sure to draw out the rod carrying contact point far enough to insure the self-centering action of the springs is brought into play. After the variation has been noted, the mean diameter of the cylinder in inches and thousandths of an inch is found by transferring to an outside micrometer. Slant the gage when removing it from the cylinder. Do not snap it out. This method has not only the advantage of being absolutely accurate from the mechanics' standpoint—as well as being exceedingly simple and quick—but also, when reboring, regrinding or honing, or when new pistons are necessary, leaves nothing to the imagination or faith of the car owner. He can see for himself the condition of the cylinders to the one-thousandth part of an inch.

Another method of gaging and testing cylinder bores, in which, however, considerable dependence must be placed on "feel", is to use an inside micrometer. (See pages 18 and 19.)

The indicating and operating parts of this instrument are very similar to those of the outside micrometer previously described and the method of reading is the same. It should be noted, however, that the inside micrometer obviously has no zero, the minimum being the length of the rod inserted

Fig. 87—Cylinder Gage (Starrett)

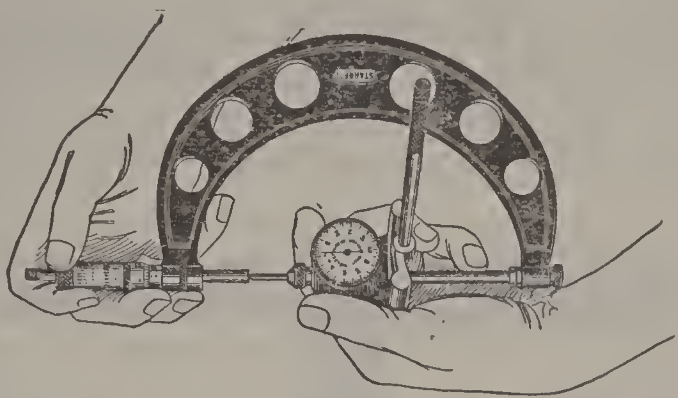


Fig. 88—Transferring Cylinder Measurement from Cylinder Gage to Outside Micrometer

in the micrometer head, as 2.000, 3.000 or 4.000, etc., inches. The rods, just mentioned, remain fixed in the sleeve of the micrometer head and replace the spindle of the outside micrometer. Otherwise, except for physical form, the tools are the same.

The anvil is placed on the outer end of the thimble and when the micrometer is opened is forced out until the end of the rod comes in contact with one side of the cylinder wall and the anvil with the other. The distance between the two is the bore of the cylinder and is indicated by the graduations on the sleeve and thimble.

When using the tool, the micrometer is held in one hand, the end of the rod brought into contact with one side of the cylinder wall and the thimble turned by the fingers of the other hand, opening the micrometer until

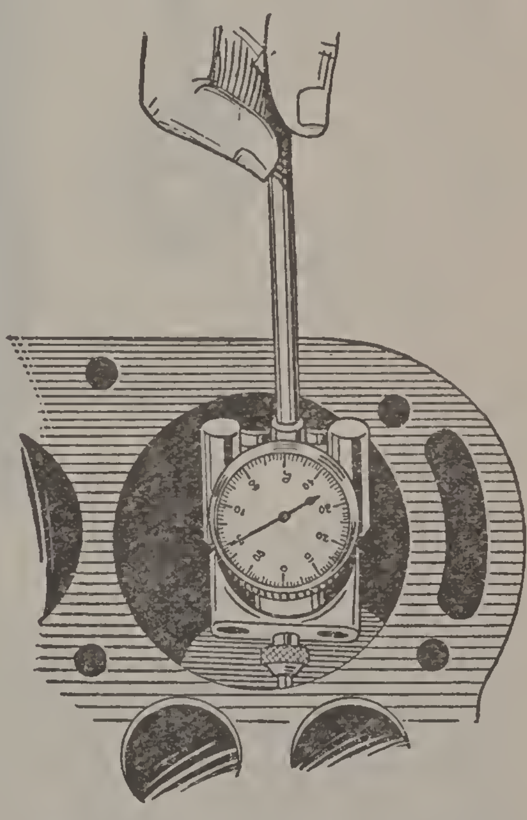


Fig. 89—Dial Test Indicator Used on Cylinder Regrinding Job

the anvil "feels" the opposite wall. Swinging the tool in an arc about the axis of the cylinder determines whether or not the bore is out of round, while by raising or lowering the micrometer, the cylinder may be tested for taper. In all cases,

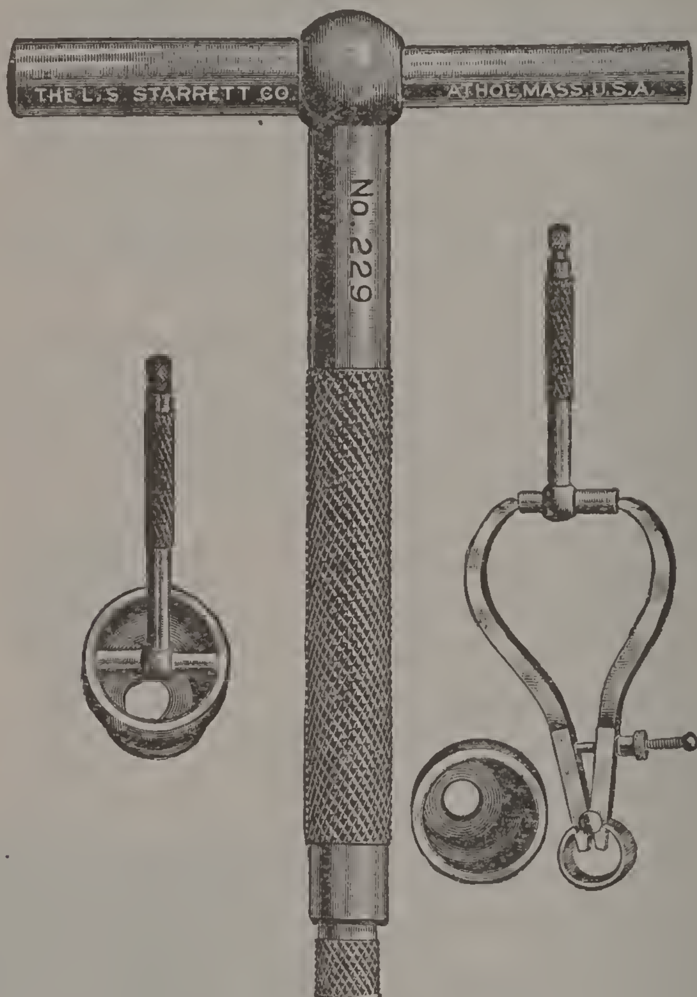


FIG. 90—TELESCOPING GAGE—STARRETT

the accuracy of the measurement depends largely upon the "feel". For measuring long bores, or holes too small to permit the insertion of the hand, an extension handle is provided.

Another tool used in a similar manner to measure cylinder bores, and which is more nearly automatic in its action — though it is not direct reading and, therefore, requires the use of an outside micrometer — is the telescoping gage.

This tool consists of a steel rod sliding in a sleeve, from which it is automatically forced outward by a light spring, and which may be locked at any point by a slight turn of the knurled screw in the extension handle which serves to lower the gage into the hole to be measured. To use the tool, all that is necessary is to compress the telescoping head and lock it in the sleeve by turning the screw in the handle. After inserting the tool in the cylinder bore, release the telescoping head or plunger. A spring under or behind the plunger forces it outward until it comes in contact with the opposite side of the cylinder wall. After turning the screw again to lock the plunger, withdraw the tool and transfer the measurements to an outside micrometer as was done with the cylinder gage.

Both the inside micrometer and the telescope gage can be used in fitting pistons to cylinder bores as well as for fitting cylinder bores to pistons. When it is desired to fit a piston to a cylinder, it is necessary only to caliper the piston with an ordinary pair of outside calipers and transfer the measurement either to an inside micrometer — where it can be read direct — or to a telescoping gage and thence to an outside micrometer for direct reading when opportunity arises or at the convenience of the machinist. However, for rapidly and accurately determining the exact condition of cylinders, a cylinder gage, such as was shown in Fig. 87, should be used.

CYLINDER GRINDING

Before starting to rebore or grind cylinders, the greatest pains must be taken to make sure that the cylinder bore is at right angles with the base of the block. To do this, the block should be mounted on angle irons bolted to the table of a good cylinder regrinding machine. In the spindle of the grinder mount a dial test indicator (See Fig. 102, page 137) and bring the point of the indicator in contact with the face of the block, setting the dial at zero. Then move the cross slide until the entire length of the block face has been traversed by the indicator point, which should read at zero

throughout. If there is any variation it indicates that the block is not properly set, and must be readjusted.

The next operation is to examine the cylinder bores, the cylinder with the greatest amount of wear being ground first, as it determines the size to which the other cylinders are to be ground. The first step is to measure or "mike" with an inside micrometer the cylinder showing the least amount of wear—usually the one at the rear of the block. This determines whether or not the cylinders have been previously reground or rebored, and, if not, it indicates the original diameter of the bore within .001 and .002 inch. Having determined the original bore of the cylinders and set an outside micrometer to this dimension, the pistons should be "miked" to determine their condition.

The condition of each cylinder bore is the next thing to be ascertained and to do this each should be tested with a dial cylinder gage set so that it would have read zero in the original bore. The information gained from the use of the cylinder gage enables the machinist to know exactly how much to remove from each bore to straighten it up and bring it into round and also what diameter oversize pistons and rings are needed. As has already been said, the cylinder which shows the greatest wear determines the size to which the others are to be ground and a standard oversize should always be selected. According to S. A. E. practices, there are four standard oversizes for cylinder bores, as follows:

1st Oversize $= d + .010$ in.
2nd Oversize $= d + .020$ in.
3rd Oversize $= d + .030$ in.
4th Oversize $= d + .040$ in.
$d =$ orignal diameter of cylinder

The grinding and regrinding of automobile cylinders is so extensive that a number of machines have been developed especially for this purpose. While the machines differ in details, the principles of all of them are about the same,

The cylinder block to be ground is mounted on a jig which holds it in exact position after the workman has properly lined it up. The grinding wheel is quite small in diameter, about 1½ or 2 in., and revolves at a very high speed, some, 3000 or 4000 R. P. M., depending on the diameter. The shaft of the wheel is mounted in a sort of eccentric whose distance off center is adjustable by means of a control lever. This eccentric revolves also, but at a very much slower rate than the wheel. By setting the machine up properly, the wheel is made to slowly turn around and revolve at the same time, its surface just touching all parts of the diameter of the cylinder. The wheel is fed into the cylinder bore automatically, the feed being adjustable.

It is usual to take several "cuts" or bites in refinishing a cylinder and these may vary anywhere from .001 to .006 in. each. The first bite is light so that the alignment of the cylinder can be observed and checked. The roughing cuts are heavy and the final finishing cut is about .001 or .002 in.

It might be thought that the cylinder refinished by the grinding method would be tapered owing to the fact that the wheel wears down as it grinds. This wear, however, is very slight and the finishing cut is made both in and out so that any difference in diameter is scarcely distinguishable with the most delicate measuring instruments. A diagram of the motions of the cylinder grinder is shown in Fig. 91A.

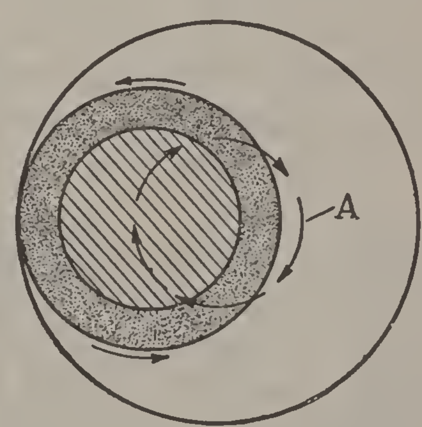


Fig. 91A—Diagram of the Motions of the Cylinder Grinder

While the standard oversizes are 10, 20, 30, and 40 thousandths inch, many grinders work in between these standards if the cylinder can be brought true with less grinding. It is not absolutely necessary that all the cylinders measure

the same exact diameter because the very slight increase in combustion space is more than offset by many other inequalities in the engine such as uneven cylinder head castings, uneven cam contours and unequal valve tappet clearances.

LAPPING CYLINDERS

Where cylinders are not more than .003 in. out of round or tapered to that extent, they can be trued up by lapping. The process of lapping is quite old and is performed with a piston which is slit with a hacksaw as shown in Fig. 91B. The handle is moved up and down and at the same time given a spiral motion. At each stroke the position of the handle is advanced slightly so that the lap gradually turns around in the cylinder. This is to distribute the grinding uniformly around the cylinder and prevent rifling or wearing a spiral groove in one spot. The lap is kept charged with fine grinding compound mixed to the proper consistency with oil. The lap is kept operating until measurements with the inside micrometer or dial gage show that the bore is true as to roundness and lack of taper. The lapping compound is then thoroughly washed out of the bore with gasoline and the new piston rings fitted. It is sometimes necessary to lap the new piston lightly in the cylinder to finish the fitting.

Lapping to remove a considerable quantity of metal is a slow and laborious job and it takes from 8 to 12 hours to properly lap a four-cylinder engine, depending upon how far out of true the cylinders were to begin with. There are on the market several electrical and mechanical lapping machines which will do the work just as well as it can be done by hand and in a much shorter time.

CYLINDER HONING

Cylinder Honing is a cylinder finishing process that has come into rather general use quite recently. The hone consists of a steel frame or cradle which supports four grinding stones which are pressed against the cylinder walls by

springs. A suitable lubricant is used and the hone is revolved in the cylinder, being moved up and down at the same time, somewhat after the manner of the cylinder lap.

Hones are generally provided with two or more grades of stones for fine or coarse honing and very often the hone is revolved by means of a portable electric drill although a drill press can be used if the cylinder block 's off the engine.

The cylinder hone can be used for truing up cylinders that are out of round within certain limits and also cylinders that are tapered or bell mouthed within certain limits. Just

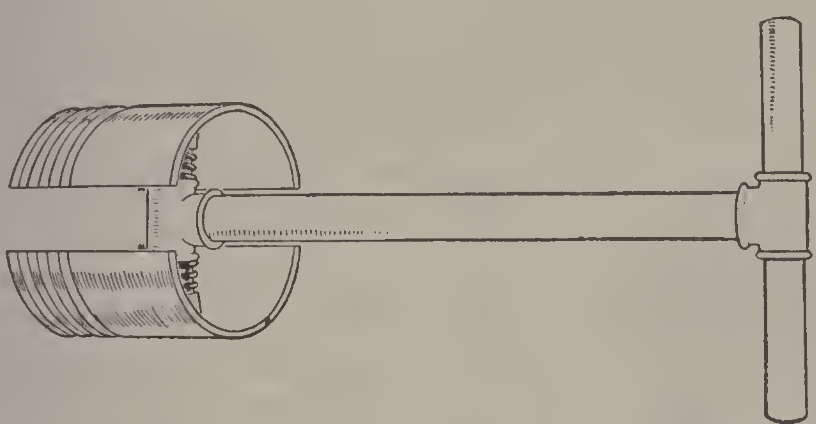


Fig. 91B—Split Piston Lap

what the limits are is a little bit difficult to say. The process is so new that some manufacturers make claims that may or may not be proved in practice. It is safe to assume that irregularities of .005 in. can be removed with a hone and perhaps more. In most cases the cylinder hone is used as a final finish after boring or sometimes after grinding. Cylinder grinders claim that this is not necessary. Although several automobile factories are using the honing process, the greatest use of the hone, however, is in the automobile repair shop.

Honing or Lapping Tools will not correct error in axial displacement. When this has occurred reboring or regrinding is the only remedy.

PISTON FITTING

Fitting pistons to oversize cylinder bores requires perhaps more care and accuracy than any other common operation in automobile repair work because, once improperly fitted, the error cannot be corrected except by complete refitting.

Having ground all the cylinders to the new size, the pistons are ground to the correct oversize, allowance being made for proper clearance.

When grinding oversize pistons there are many factors to be taken into consideration in determining the proper clearance, such as: the material from which the piston is made, the make of piston, design and construction of the block and the use to which the motor is to be put when the job is complete. The common rule is to allow .001 inch clearance at the skirt of the piston for each inch of piston diameter. This varies, however, according to the material from which the piston is made, its design, etc. For example, aluminum alloy pistons require much more clearance than cast iron, because aluminum alloys expand much more than cast iron. On aluminum pistons the customary allowance is about .003 to .004 in. per inch of bore. On the other hand, constant clearance types of pistons, with slit skirts, do not require any more—and in some cases, less clearance than cast iron. The clearance of the piston tapers from the head to the lower edge of the skirt, a 3 inch piston having .003 in. clearance at the skirt, requiring .009 in. clearance at the head, and .006 in. at a point just below the rings. See Fig. 92.

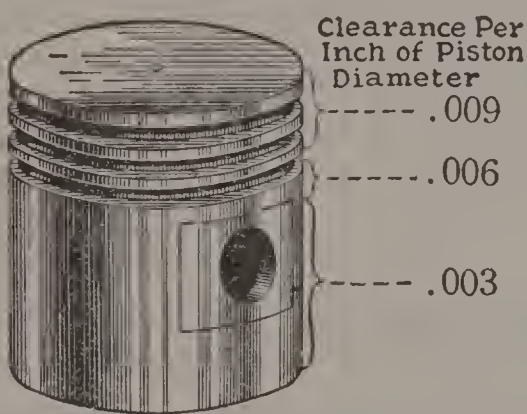


Fig. 92—Clearances to Allow at Different Points on Piston

Care must be taken also to see that the pistons square up

with the connecting rod, a piston aligner being used in this operation.

It should be noted that standard oversize pistons come approximately .080 inch oversize—an amount altogether too great to remove quickly and efficiently by grinding. The better way is to turn the pistons down to within .015 in. of the finish size and then complete the operation on a piston grinder. In most cases, pistons should be backed off, or relieved at the top.

While, generally speaking, piston clearance for cast iron pistons may be taken as .001 in. of clearance at the head of every inch of bore diameter, clearances for taxicabs, racing cars, trucks and tractors should be from .00125 to .005 in. greater and the same generous allowance should be made whenever there is reason to believe that the motor is likely to be overworked or carelessly driven.

FITTING PISTON RINGS

Fitting the rings to new or oversize pistons is every bit as important an operation as fitting the pistons themselves, or grinding the cylinder bores. No motor can have either long life or full efficiency unless the complete piston assembly is made with the greatest care and accuracy.

Piston rings may be divided into three general classifications—the plain ring, the step-joint, and the "oil-proof" ring. There are also the so-called spiral cut rings. The difference lies mainly in the kind of joint and types are shown in Fig. 93. Each style of ring has its advantages

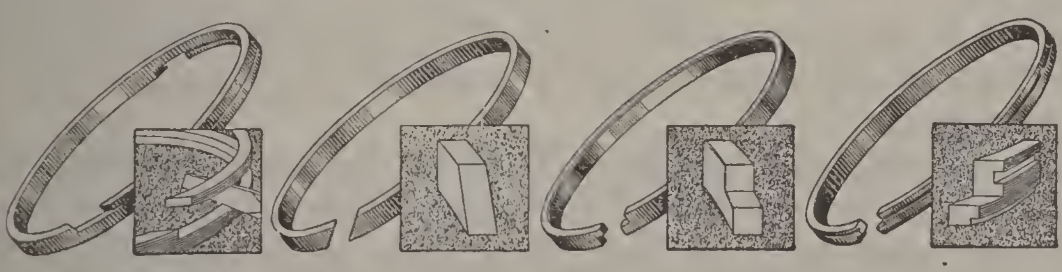


Fig. 93

and disadvantages, and each has a considerable group of good repair men who prefer it. Whatever ring is used, care should be taken to see that it is perfectly round when compressed, so that it will bear rightly and evenly at all points in the wall of a reground cylinder.

Piston rings fit into grooves cut in the wall of the piston (See Fig. 92) and while it is impractical to try to work to any measured clearance, the rings should lie just free in the grooves, or so that they can be turned around the piston with the fingers. It may be that, when fitting rings to new pistons, the grooves are a trifle narrow and in that case should be touched up on the lathe, using a fine file which has one blind or smooth edge so as not to deepen the groove. The proper depth of the ring groove is 1/64 in. greater than the thickness of the ring. If the grooves are badly worn, however, use an overwidth ring and true up the grooves in a lathe or by means of a hand piston regrooving tool. Be careful not to try to fit too large an oversize ring in a cylinder as it will assume an oval shape under compression, bearing on the cylinder wall only at the ring gap and a point diametrically opposite and will leave openings for the gas to blow by. Be sure that the ring bears tightly and evenly all around the wall of the cylinder and try it for the full length of the cylinder to be sure it will not stick when traveling with the piston.

Clearance in piston ring gaps should never be less than .005 in., the general practice being to make it .0015 inch per inch of cylinder bore with the exception of the top ring which should have .003 inch gap clearance per inch of cylinder bore, or just twice the amount allowed in the other rings. The width of the gap may be readily checked by means of a thickness gage.

Sometimes new rings are fitted to old pistons and cylinders to avoid the cost of a regrinding job. The practice cannot be recommended from any standpoint save that of sheer necessity, but where followed, rings .005 to .010 oversize may be used.

In some cases, oil pumping, or an excessive consumption of oil, can be corrected by beveling the lower outside corner of the bottom groove in the piston and drilling 1/16 inch holes at an angle, and at equal intervals, through the bevel so that the oil will run back into the crankcase.

FITTING PISTON PINS

In fitting piston pins great pains should be taken first of all to make sure that the connecting rods are neither twisted nor bent (See Fig. 94), and that the axis of each cylinder bore is exactly at right angles with the axis of the crankshaft as was explained under Cylinder Grinding, pages 107 to 110.

Where no piston aligning jig is available, a piece of shaft the same diameter as the crankshaft may be placed in the big end of the rod, the latter held mechanically in a vertical position and the squareness of the rod and crankshaft tested by means of a square placed on the shaft and on either side of the piston.

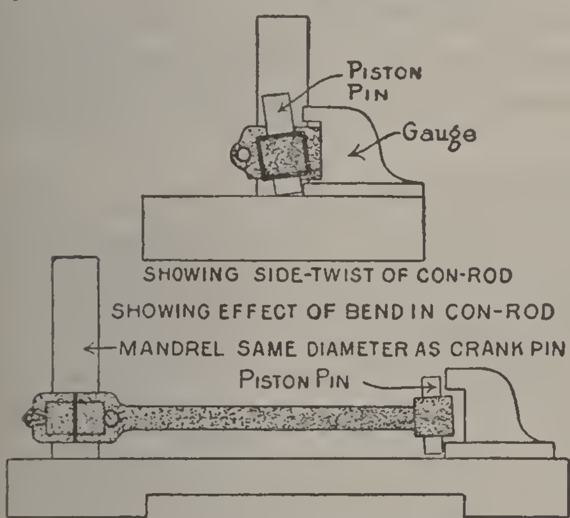


Fig. 94—Testing Piston Pin and Rod Alignment

The next step in the work, after having ascertained with an inside micrometer—or by other means—the oversize pin to use, is to ream the bushings in the connecting rod or piston wall—according to the design of the motor—in which the piston or wrist pin turns.

When reaming a split-end rod oversize for an oscillating pin, shim the split, draw the clamp bolt up tight and ream until the pin just slips in. When reaming bushed rods allow from .0025 to .00075 inch clearance for oil film. A piston pin should be so fitted that it will just hold the weight of the connecting rod and no more.

When reaming a piston using the anchored type of pin, ream the piston boss opposite the set screw boss so that the pin will be a rather tight pressed fit, but be careful that it is not fitted so tightly as to run the risk of splitting the light metal of the boss.

With oscillating pins the cast iron piston bosses are the bearings and a clearance of from .005 in. for a small pin to .001 in. for a large pin should be left. In reaming bronze bushed pistons the same clearance should be left as for a connecting rod bushing.

Before fitting aluminum or other alloy pistons with pins, the pistons should always be placed in a tub of boiling water to heat them to about 200°, at which temperature the pins are fitted so they are just free.

FITTING CRANKSHAFT AND CONNECTING ROD BEARINGS

Fitting crankshaft and connecting rod bearings is a fairly common operation in automobile service stations and repair shops and one of great importance and influence on the efficiency of a motor.

Bearings are commonly fitted in one of two ways. The older method is known as "scraping-in" bearings and while it is today the common practice in many shops—especially where machine equipment is lacking, or the man in charge of the work is a thorough machinist and came into automobile work from the jobbing machine shop trade—the present tendency is toward "burning-in". There are perhaps two main reasons for this change. "Scraping-in" is entirely a hand operation, consuming much time, and it is difficult to charge properly for this work in the face of what are called "factory flat-rate, time-limit charges" for operations of this sort. In order to compete successfully with the "flat-rate" charges set by manufacturers' service stations and factories on this work, methods such as the factories themselves followed had to be adopted. The other reason is that hand scraping,

technically, does the very opposite of what is desired, for it lifts the metal instead of packing it down.

Whichever method is followed—scraping or burning-in—the preparatory work is the same. The crankshaft should be tested for alignment, crankpins and main bearings trued up and all worn fillets restored.

A very satisfactory method of restoring fillets is to braze on new material with an oxy-acetylene welding torch and phosphor bronze filler rod of good grade. The fillets should then be machined either on a lathe equipped with offset centers (Fig. 96), or on a grinding machine having the proper centers for holding this type of work.

If the crankpins are not too deeply scored, they may be

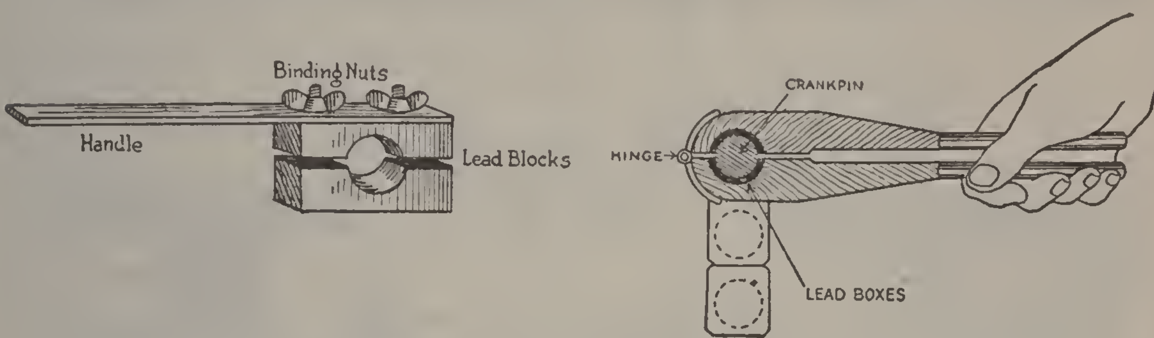


Fig. 95—Hand Lapping Tools for Crankpins

trued up with a fine file and emery cloth, or better, a hand lapping tool such as is shown in Fig 95, and which consists roughly of two lead blocks about half as wide as the distance between the webs of the crankshaft and turned concave so that when brought together, either by clamp screws or by pressure on the handles, they will bear firmly and evenly on the surface of the crankpin. An abrasive paste of fine emery powder and oil is used to coat the lead blocks before attaching the tool to the crankpin. When using the lapping tool, the proper method is to put the crankshaft between centers on a lathe, as in Fig. 96, though where a lathe is not available, the crankshaft may be clamped to the workbench and the lapping tool turned by hand around

the stationary crankshaft. If the crankpins are worn out of true to any great extent, they must be turned or ground true, and then lapped if a really first class job is desired. Obviously, the setup of the crankshaft in the lathe or grinding machine will have to be changed as the different crankpins are being machined, so that the pin on which the work is being done

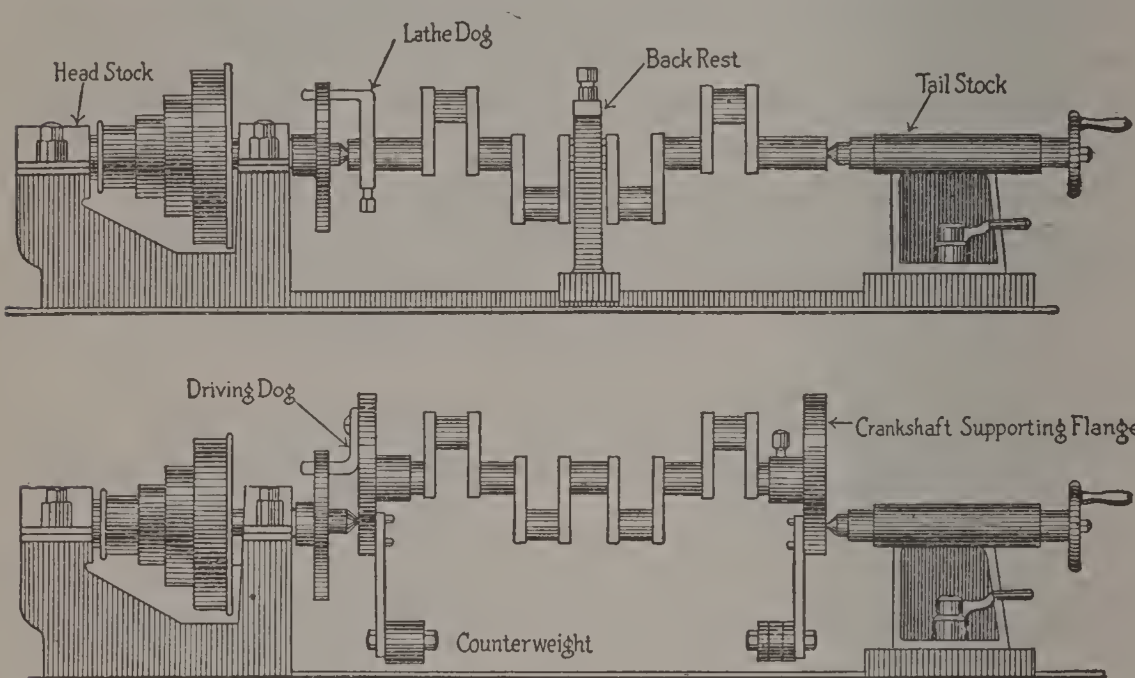


Fig. 96—Methods of Placing Crankshaft Between Lathe Centers for Truing Main Bearings

will revolve on the main center line passing through the lathe centers.

Having trued the crankshaft the next operation is to fit the main bearings to the shaft journals. After having tested—and if necessary trued up and aligned—the connecting rods, the bearing caps should be trued, using a surface plate and Prussian blue, and then dressed down until, when clamped on the crankshaft, they grip tightly and require shims to make it possible to turn the shaft. The shim thickness required should be about .002 in. and may be ascertained by feeling with a thickness gage under both sides of the cap. Care should be taken, and a pair of calipers used to check

the outside diameters during this operation, that surfaces remain parallel so that the bearings do not grip only at one end. The connecting rod caps should be treated similarly, bringing the work to the point where the bearings themselves are to be fitted, either by scraping or burning-in.

In scraping-in bearings considerable time and handling of the crankshaft may be saved by attaching the crankshaft to the bench, clamping each pair of main bearings together with a suitable clamp and doing the preliminary scraping on one bearing at a time. As soon as the "roughing" is accomplished the final scraping-in should be done with all the bearings in place, the crankshaft being in position in the inverted upper half of the engine base, and the crankshaft revolved to determine the area of seating.

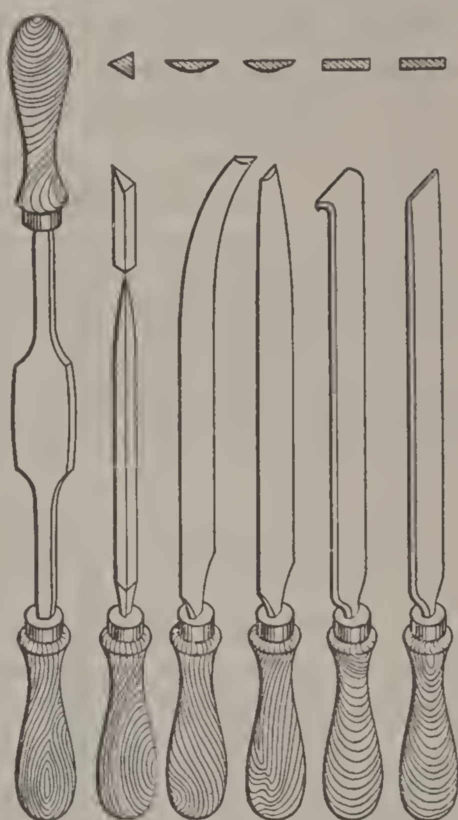


FIG. 97—BEARING SCRAPERS

Scraping-in bearings is slow and tedious work and requires considerable patience and care. The entire surfaces of the crankpins are smeared, not too heavily, with Prussian blue. When the bearings are put in place, drawn together, and the crankshaft revolved a few times, the high spots on the bearing are indicated by smears of Prussian blue wherever the bearing came in contact with the crankpin. These high spots are removed by means of bearing scrapers (Fig. 97) and the bearings again tested in the crankpins in the same manner as before. This is repeated until the entire inner surface of all the bearings seat on the crankpins as indicated by a film of Prussian

blue spread evenly over the inner surfaces of the bearings. Before the scraping is started, oil grooves should be cut in the bearing with a round-nosed chisel. Do not cut oil grooves too deep nor carry them to the extreme edge of the bearing. Connecting rod big end bearings are scraped-in in the same manner as main bearings.

In burning-in bearings, the operations are the same up to the point where actual scraping begins under the other method, the scraping being avoided by placing the crankshaft assembly in a burning-in machine. When clamping the work in place be sure that the main axis of the crankshaft lines up with the axis of the driving spindle on the burning-in machine. While burning-in do not revolve the crankshaft faster than 100 R. P. M. and use plenty of oil or, as is some times recommended, revolve the shaft at about 250 R. P. M. using no oil up to the point where the skin of the bearing begins to melt and forms clear around the shaft. At this point the machine is stopped, the bearings heavily oiled

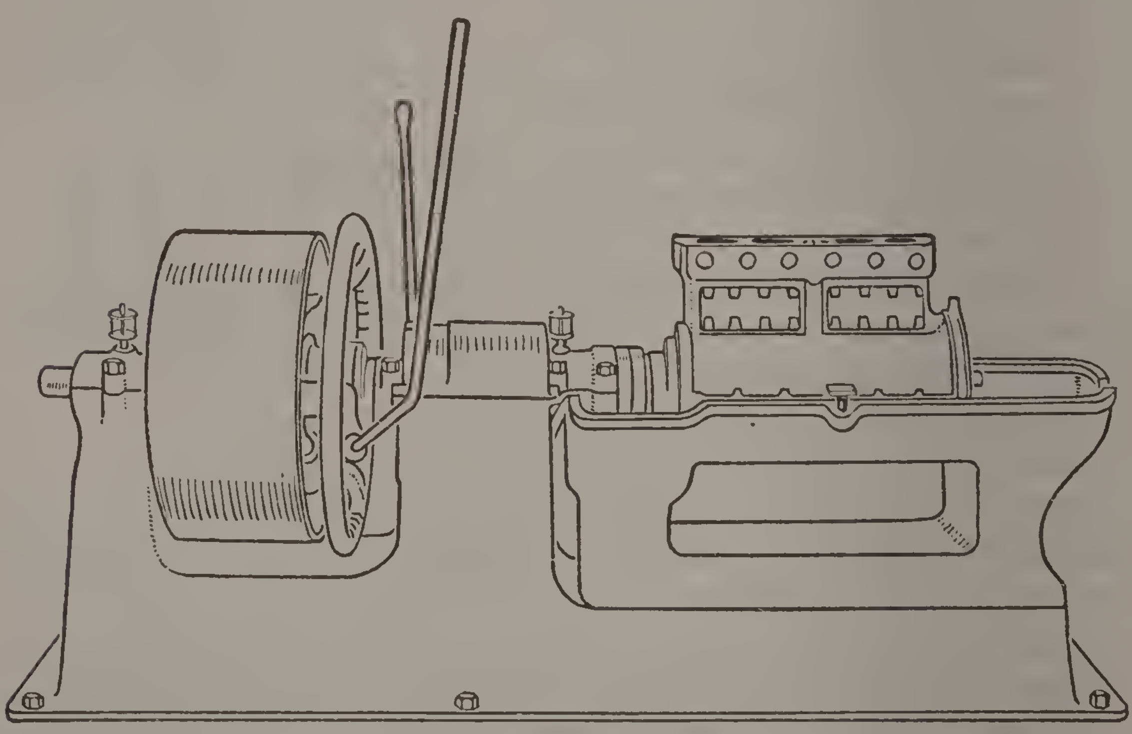


FIG. 98—BURNING-IN MACHINE

and the bearings run in under power until they become well burnished.

When the bearings become heated they should be allowed to cool and then removed and inspected for bearing surface. When the bearing surface is less than 90% of the total surface of the bearing a thin shim should be removed, or the cap dressed down, and the bearing again burned-in. On crankshafts with forced feed lubrication—or shimless main bearings—there should be a little looseness (.002 in. approx.) between the cap and the seat to permit an oil film to form between the pin and its bearing while burning-in.

VALVE FITTING

Valves of cars which have seen much service should be very carefully examined before any attempt is made to refit them, as often it is cheaper and better to replace the old valves with new ones. Both valves—including the valve stems—and valve stem guides should be carefully examined for wear. Any valve stem that is worn or which has as much as .008 in. clearance in the guides indicates a need for a new valve, a new guide, or both. On every valve stem there is a small section which never enters the guide and is therefore never worn. Before grinding an old valve, it should be tested between centers to see if it runs true on the worn section of the stem. If it runs true, the valve may be ground either between centers or while held in a collet chuck which grips the unworn section of the stem. When the valve does not run true on the worn section, it should be ground while held in a collet chuck which grips the worn section at a point that will average up the wear, so that the valve heads will more nearly fit the seats when the worn stems are replaced in the worn guides. In most cases, however, a valve whose stem is worn badly enough to require this latter method of grinding should be replaced. In examining the heads and stems, a micrometer should invariably be used.

Warped or bent stems are seldom, if ever, worth straight-

ening, because in most cases the valve stem guide has been worn out of true. However, when no new valve is available, the old stem may be straightened by placing the valve between lathe centers and crowding the stem over with a tool rest until it runs true. They should, however, be exchanged for new valves at the first opportunity, for a properly fitted valve has but .0025 to .003 in. clearance in the guides and it is almost impossible to straighten bent stems so that they won't bind.

Valve faces and seats, improperly ground or pitted, should be refaced before grinding in, as should also be done in cases where the valve head is warped. In refacing valve seats a reseating cutter should be used, the edges of which are at the proper angle for the motor on which the work is done. The stem of the reseating tool should be the same size as the valve stem in order to keep the tool as rigid as possible. In refacing a valve seat take off no more metal than is absolutely necessary. Valve seats that are too wide may be reduced by counterboring the valve chamber, leaving a seat from 1/16 to 3/32 in. wide. Counterboring is preferable to boring because the latter method often prevents the use of standard diameter valves. For refacing the valve it should be placed in a regular valve grinding machine or—if that is not available—the stem end should be held in a universal chuck on the shop lathe, with the tailstock center of the lathe in the countersink on the head of the valve. The job may then be done with a tool-post grinder set at the proper angle, the valve rotating at low speed, and the grinder moved back and forth by the compound rest feed screw. Again, be careful to remove no more metal than is absolutely necessary.

The valve stem guides should now be examined and if new valves with oversize stems are being put in, should be either reamed or replaced. New valve stems should be lapped into the guides in the same manner as a new piston is fitted. When the weight of the valve will cause it to move very slowly to the bottom without stopping until the valve head

rests on the upper end of the guide, it may be assumed that the fit is correct. In reaming valve guides, care must be taken to keep the reamer central. Use a very light pressure so that the reamer will cut its new path rather than follow the old hole.

The next step in operations is the grinding-in of the valve on the valve seat. In some of the smaller shops, not equipped with proper valve grinding machines, "grinding the valve" and "grinding-in on the valve seat" are synonymous. The better way, however, is to "grind the valve" on a machine designed especially for that work and to view the "grinding-in on the seat" more as a lapping operation and one that is often unnecessary if the valve has been properly ground on the machine and the seat reamed with a valve-reseating tool or reamer that was ground on the same grinding machine and to the same angle as the valve. However, when the "grinding" is of necessity a hand operation, a coil spring should be placed over the stem and under the head of the valve (Fig. 99), so that the valve head is about an inch above the seat, which is well coated with grinding compound, and the valve turned in the seat by means of either a hand or electric valve grinding tool. After every few turns the tool should be lifted enough to shift the position of the valve. After rough grinding the operation is repeated, using a very fine grinding compound until the entire surface of the valve face is silvery in appearance. Both valve and seat should then be thoroughly cleansed of all traces of the grinding compound and the valve permanently inserted. Some machinists finish the job by lapping the valves in with kerosene, which gives a high polish to the seats and tends to prevent carbon deposit. The seating of a valve may be tested by coating the seat with Prussian blue just as is done in fitting main and connecting rod bearings. While grinding the valves the valve ports should be stuffed with waste

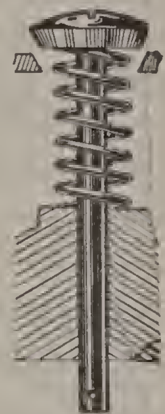


Fig. 99—Ready to Grind-in Valves

or rags to keep the grinding compound from getting into the valve guides or cylinders.

It is generally considered good practice to decarbonize the motor—either by scraping or burning-out—whenever the valves are ground, because once the carbon crust is disturbed, it will continue to flake off and will work under the valves.

Valve tappets should be adjusted when the engine is warm, using a thickness gage and allowing from .002 to .005 inch clearance, according to standards recommended by the various manufacturers. Valve clearances should be adjusted while each cylinder is on its compression center. If the valve adjusting screws have had hollows pounded into their ends they should be ground flat. This may be done by placing them in the chuck of a drill press and running them against an oil stone lying on the drill press table, or by grinding them in a proper grinding machine, such as was used to grind the valves.

LAPPING BEARINGS

As has already been noted, bearings often have their fitting completed by lapping. This consists in putting a special bearing lapping compound in the bearing and lapping it to the shaft. This works very well with bronze bearings, but care must be taken to see that the right kind of compound is used with babbitt because there is likelihood of the compound imbedding in the soft babbitt and thereafter gradually grinding the steel shaft down. A compound that breaks down and disappears or dissolves is necessary for use with babbitt.

ANTI-FRICTION BEARINGS

Ball and roller bearings need no fitting. The inner and outer races are finished to close limits and replacements are made by driving out the old parts and driving in the new parts. Care must be used to drive straight so as not to cock the race. Roller bearings of the flexible roller type have an outer shell which is split and therefore requires little

pressure to remove or replace. The rollers usually operate directly between this shell and the shaft. Adjustment is possible in some types of anti-friction bearings and in some other types there is no adjustment.

FITS AND FITTING*

Various kinds of fits are used in automobile construction. In some cases the fit is fairly loose, or has clearance so that the parts are free to move on each other. In other cases the fits are made very tight and are called driving fits.

Running Fits

Parts that are to move on each other at varying speeds are given sufficient clearance—known as a running fit—to enable them to perform their duties with a minimum of friction. The kind of metal and degree of heat attained by the part under running conditions determine the amount of clearance. Some examples of running fits are the main and connecting rod bearings, camshaft bearings, pistons, cooling fan, rear axle and propeller shafts (the running fits being in the ball, roller or bronze bearings), and the front and rear wheel bearings.

Driving Fits

Some parts require a driving fit against another part. Practically all the solid bronze bushings such as used in the steering spindles, piston pins, reverse shaft and propeller shaft front bearings are driven or forced into machined openings. The outer races of ball and roller bearings are a driving fit in the wheels, castings or wherever they go and the inner races are a driving fit on the shafts. The balls or rollers are a running or really rolling fit between the two races.

Shrink Fits

There are a few places on the automobile where a shrink fit is used. In some cars the flywheel starter gears are made

* See also pp. 29–33, Vol. I, Starrett Books, and pp. 14–18 and 152–154, Vol. II, Starrett Books.

as separate rings which are heated to expand them, slipped over the flywheel and cooled. As the fit is made very close in machining, the ring shrinks on the flywheel when it cools and is immovably fixed if the work has been properly done. Damaged flywheel gears are quite often replaced by turning off the old gears and shrinking a new gear on.

Taper Fits

Several parts of the automobile are fitted with a taper, key and nut. The inner and outer tapers are accurately machined and the key is usually of the Woodruff or half-moon type. By tightening the nut on the shaft the part is driven fast on the tapered shaft and also prevented from turning by the key. The pinion gear is fastened to the rear of the propeller shaft or pinion shaft in this way and the rear wheels are fastened to the axle shafts in the semi-floating type of axle in the same manner.

BENCH WORK*

Bench work includes laying out, chipping, filing, polishing, hand reaming, hand tapping, and all the many shop jobs done at the bench or in a vise.

LAYING OUT

This is the shop term which includes the placing of lines, circles, and centers upon curved or flat surfaces for the guidance of the workman. It is somewhat analogous to mechanical drawing. It differs in one important respect, however, that while a line drawing is seldom scaled and therefore exact accuracy of spacing is not required, in laid-out work, the lines, circles, centers, etc., are to be followed exactly. All lines, centers, etc., should, therefore, be exactly located and placed, and all scriber, divider, and center points should, while in use, be exact and sharp. Particular care must be maintained to insure fine and accurate laying out.

* See also pp. 35–42, Vol. I, Starrett Books. Also pp. 52–60, Vol. II, Starrett Books.

Preparing the Surface

If no work of special accuracy is desired, carefully rubbing chalk, or white lead mixed with turpentine, upon the surface of the work will be sufficient as a coating. For fine exact layouts a special marking solution must be used. The one in common shop use is a mixture of one ounce copper sulphate—or blue vitriol—to four ounces water. A little nitric acid may with advantage be added. This solution applied to a cleaned iron or steel surface gives a dull coppered surface, and the finest line scribed upon it is brilliantly visible. Still another method used on die, or other extremely fine work, is to heat the piece to a blue before scribing.

Scribing Lines

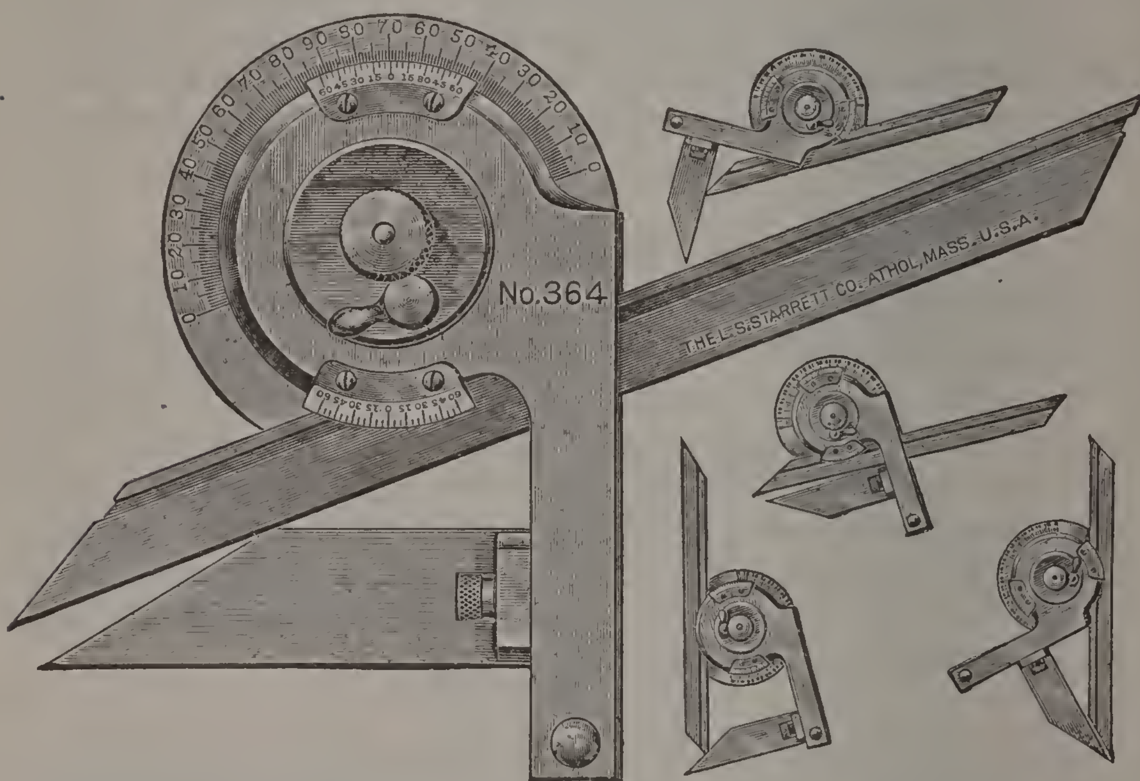
The usual scribing points are those common to dividers, hermaphrodite calipers, scratch awls, scratch gages, surface gages, and trammel points. Combined with the scribing points may be used steel rules, bevel protractors, steel squares, steel straight edges, levels and measuring rods, micrometer or vernier height and depth gages, and the various center punches. Ability to so combine and make use of the various tools as to insure accuracy is a considerable asset to the man who does laying out.

Protractors

As made for machine-shop use the common protractor is provided with attached straight edges, and can be used either to measure or to lay off lines at an angle to each other. Measuring the angularity of two or more lines with a protractor is termed "reading the angles". As oftentimes its use is determining the angle made by two surfaces (a bevel), the tool is usually termed a bevel protractor. Protractors for common shop use are graduated to degrees through a length of circumference of one hundred and eighty degrees. An attached vernier enables the user to read angles to one-twelfth of a degree (five minutes).

Laying-Out Plate

If desirable results are to be obtained in laying out flat work, special metal plates upon which to rest the work and the tools must be provided. These are known as leveling, surface, or laying-out plates; they furnish an accurate plane surface upon which work and tools may be placed. The



UNIVERSAL BEVEL PROTRACTOR WITH VERNIER AND ACUTE ANGLE ATTACHMENT

size of these plates varies from those of small areas used in laying out small jigs, etc., to those for large pieces, having sides several feet in length. The work may be laid directly upon the surface of the plate or held upon leveling strips, blocks, or parallels placed on the plate. In other cases it is convenient to clamp the work to knee or angle irons, which are then placed upon the leveling plate.

CHIPPING

Formerly many of the surfaces of machine parts were hand-chipped and filed to a fit. While the mechanic in the modern shop can usually find methods of machining most of the surfaces he needs to fit up, there are still occasions when the work has to be hand-chipped.

Tools Used

The common chipping tools are a hand hammer and a cold chisel. The hand hammer should weigh not less than

FIG. 100—CHIPPING

three-quarters of a pound nor over two pounds, and may be either of the ball pein or flat pein type. A chipping hammer should balance well in the hand when fitted to a handle not more than sixteen inches long. The handle near where it enters the hammer should be thinned and worked down to a shank that is somewhat flexible, so that the shock to the arm and hand will be less. The face of a good chipping hammer should crown slightly.

Chipping chisels, ordinarily termed cold chisels, are of various sorts, and are often known by the shape of the cutting end; for example, flat, cape, roundnose, diamond, and gouge chisel. The steel from which they are made should be eighty to ninety point carbon, of octagon cross-section, with the cutting end forged to the desired shape, well packed by the forge hammer, hardened, and the temper drawn to a blue. The hammer end of the chisel should be forged from the octagon to a reduced round, but not hardened. Flat-chipping and cape chisels should be ground with straight, symmetrical cutting edges, at as acute an angle as the nature of the work will permit.

In hand-chipping the hammer handle should be grasped near the end and the hammer swung free from over the shoulder with an easy forearm movement. Hold the chisel loosely in the hand at an angle with the work that permits an even chip of right depth. The vision should be directed to the cutting edge of the chisel, rather than at the end struck by the hammer. Avoid gripping hammer or chisel tightly, as this rapidly tires the hand and arm.

In shops which have compressed air, use is made of the modern pneumatic chipping hammer, which does remarkable work of the heavier sorts.

Cold chisels are ground on an emery or other type grinding wheel. Care must be used not to burn the steel by overheating it on the wheel. A pot of water should be kept near the wheel and the point of the chisel dipped in the water every few seconds to keep it cool. If the bright, ground

edge of the tool should turn blue from the heat any time, that part has been overheated and will have lost its temper. Such an edge will turn over as soon as it is used on work because it will be too soft.

It is necessary in such a case to reharden and retemper the tool. In hardening and also in forging, the chisel should be heated to a cherry red. If the chisel is forged when it is below a cherry red, the steel will most likely split or check, and when used later, little chips will break out. If heated beyond a bright cherry red, the steel will burn, throw off bright sparks and become pitted.

For chipping very heavy iron or steel castings, the angle of the cutting edge can be as blunt as 90 degrees or a right angle. For lighter work, brass, bronze, etc., the angle can be sharper, say about 60 degrees.

In automobile work, chisels often require a special grinding. For instance, in cutting the rivets from worn out brake linings and clutch facings, it is very much better to grind all on one side, the other side being flat like a wood chisel. For cutting oil and grease channels in bronze and babbitt bearings and bushings, the chisel is more the shape of a punch with a round sharp edge. A heavy chisel with a very blunt edge is used for cutting off bolts on wheel flanges and for cutting off rivet heads.

WELDING, ETC.

Metals may be joined together or be separated by means of heat. Joining processes of this kind are known as welding, brazing, soldering and lead burning. Welding produces the strongest joint. It may be done with an oxy-acetylene flame or by means of an electric arc, either of which are capable of melting any of the metals used in automobile construction. Welding is used for cast and wrought iron, steel and aluminum. In any case, the adjoining surfaces to be welded together are brought to the melting point and additional metal is melted from a rod of the same material and flowed over in

sufficient quantity to fill up the gap. In welding certain kinds of articles like cylinder castings, gear wheels, etc., preheating with a gas flame is necessary so that there will not be strains set up in the piece due to unequal contraction.

Aluminum welding does not require such a high degree of heat, but the metal oxidizes very rapidly and it is necessary to use welding flux to prevent this oxidation.

The possibilities of welding are almost unlimited. The process can be used for patching, building up worn parts, repairing cracks, etc. The welding flame can also be used for cutting shafts, frames, etc. Solid rubber tires and rims that are of no further use can be cut off the wheels with the torch. The odor is objectionable, but the operation is somewhat quicker than other methods.

Brazing

This is sometimes known as hard soldering and consists of joining iron or steel parts with a softer metal such as brass or bronze. A gas flame is employed, this not being sufficient to melt the iron or steel, but to bring it to a dull red heat. A flux such as borax is used to prevent oxidation and the brass solder or spelter applied and melted as desired. Brazing is often employed to repair cracks in cylinder waterjackets where great strength is not required. Brazing is also used in connection with riveting to make a stronger joint than the rivets alone would afford.

Soldering

Soldering is done with a solder composed of various parts of tin and lead. A good solder, called half-and-half, consists of equal parts of tin and lead. Radiator repairing is done by soldering, the parts of the radiator being made of brass and copper which take solder very well. It is necessary that the surfaces to be soldered, as well as the soldering iron itself, be mechanically and chemically clean. The work is,

therefore, scraped and then covered with a soldering paste of some kind. There are many such preparations that are far superior to the old, time-killed muriatic acid. There are also self-fluxing solders which are effective and quick to work with. There is an increasing tendency to solder electrical connections and joints to prevent corrosion and possible breakage.

Lead Burning

Storage batteries have their cells connected by means of lead posts and straps. When overhauling batteries it is necessary to break these connections and in reassembling the connections must be again joined. This is done by lead burning, which might be best described as lead welding, as the parts are brought to the melting point and more lead melted in to fill up the space. Certain molds for the posts and straps are necessary in carrying out the work. Besides the gas or oxy-acetylene flame, the electric arc is also employed in this work. The lead used is pure lead with sometimes a slight amount of antimony added to secure hardness.

DECARBONIZING

Oxygen by itself is used for decarbonizing cylinders. The oxygen is introduced through the spark plug holes by means of a copper tube and the carbon ignited with the flame from a match, the carbon burning completely away in the atmosphere of oxygen. The process is particularly favored on engines of the solid head type where the removal of carbon by scraping—although it is commonly recognized as better practice—would necessitate the removal of the cylinder block and the taking out of the pistons. If the pistons are kept at top position during the burning and means taken to prevent fires from the sparks, this method of carbon removal can do no damage to the car or its parts. The gasoline supply must be shut off and the carburetor and fuel lines drained before starting to burn out the carbon.

THE ACETYLENE TORCH

The acetylene torch has found a great deal of use in the automobile repair shop. Its principal function is in welding broken metal parts, but it is also very useful for heating parts to be bent and in removing the carbon from cylinders.

The apparatus required consists of a tank of oxygen and a tank of dissolved acetylene, a torch with a number of different size tips, gages for the tanks, a reducing valve and a decarbonizing tip for carbon removal. The equipment is all purchased outright with the exception of the gas tanks which are returned for refilled ones from time to time.

Welding of all kinds can be performed with the acetylene torch. Metals such as cast iron, steel forgings and aluminum can be welded and brazing or hard soldering can also be done. Where the parts to be welded are small, they are prepared by cutting a channel for the weld or otherwise preparing the edges after which the torch is played on the edges in a systematic motion until they are about to melt. The welding rod, which is of the same metal as the parts to be welded, is then melted in the flame and the parts all melt or weld together. A suitable flux is used to prevent oxidation. In welding aluminum, special care has to be taken as the metal oxidizes very rapidly, and unless the weld is free of the oxide, the strength will be reduced or the metal will not weld at all.

Large work such as cylinder blocks have to be preheated in a preheating furnace because otherwise the expansion and contraction caused by the excessive heat at one point would result in the weld cracking as soon as it cooled.

When cars come into the shop with bent axles and frames, the heat of the welding torch is very helpful in bending them into alignment again. Front axles are made with a very high factor of safety, therefore it does not do any great deal of harm to soften them in the flame. Small parts such as steering knuckles should never be heated to bend them as the heat treatment will be destroyed and the part will be so

weakened that it will be a source of danger. Neither should these small parts be bent cold as there is liable to be a fracture in the metal. The best thing to do is to replace the damaged parts with new ones.

In removing carbon from the cylinders, the gasoline is first drained from the carburetor to prevent any possibility of fire.

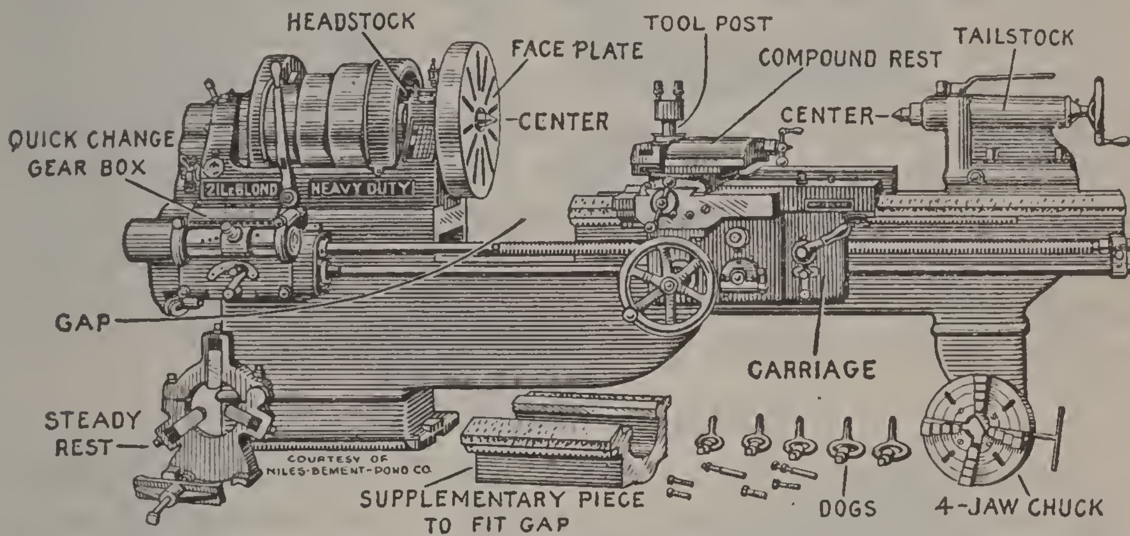


Fig. 101—Gap Lathe with Equipment for Automobile Repair Work

The spark plugs are removed and the crank turned till the pistons are at top dead center. The decarbonizing tip is then attached to the oxygen tank. No acetylene is used, only oxygen. The tip of the tube is inserted through the spark plug hole and a lighted match dropped in after turning on the oxygen. The carbon will then burn in the atmosphere of oxygen until it is all consumed. If any remains, a scraper is used to stir it up a little, after which the process is repeated.

THE LATHE*

The engine lathe is capable of producing the largest variety of products of any of the machine tool family and—because of its versatility and the almost infinite number of operations

* See also pp. 65-96, Vol. I, Starrett Books, and pp. 27-28, 107-112, 134-141, 150-151 and 156-157, Vol. II, Starrett Books.

ordinarily performed on special machines, but which can be done on a properly equipped lathe—should be part of the tool equipment of every garage or repair shop that makes any pretense to render complete service or which intends to operate at a profit.

For repair shop work a gap lathe, such as is shown in Fig. 101, is recommended. The gap in the bed makes it possible to swing much larger work—such as flywheels, etc.—than would be possible in lathes with solid beds and it will accomplish everything that the ordinary lathe will do. The lathe should have quick change gear box, screw-cutting attachments, elevating compound rests, hollow spindles, chucks, face plates, dogs, steady and back rests, etc., and automatic longitudinal and cross feeds. It should be capable of cutting any number of threads per inch from four to forty.

Care of the Lathe

Lathes, like all other machine tools, require care. Especial attention should be given to applying a suitable machine oil to all the bearings, for improper lubrication of the wearing surfaces is one of the immediate causes of excessive wear. A medium-size, flexible-bottom oil can is best for this purpose, and oiling should be frequent on those bearings which are given the severest service, either from excessive pressure or from high speed rubbing. All oil holes should be kept free and clean, and where possible should be protected from dirt. Those bearings, as, for example, the ways upon which the carriage moves, which by construction are hard to protect from dirt, should be frequently cleaned and re-oiled. At least once a week the lathe should receive a thorough cleaning, and it is recommended that the bearings be washed out with kerosene, as a plugged oil hole prevents the proper lubrication of the bearing. Never lay files, etc., on the ways of a lathe.

Indicating and Adjusting

Upon the condition of the centers rests to a large degree the accuracy of the work produced by the lathe. After

lubrication of the lathe, the centers and the tapered holes in which they fit, should be cleaned and tested. The "dead" or tailstock center should have a hardened point to resist wear.

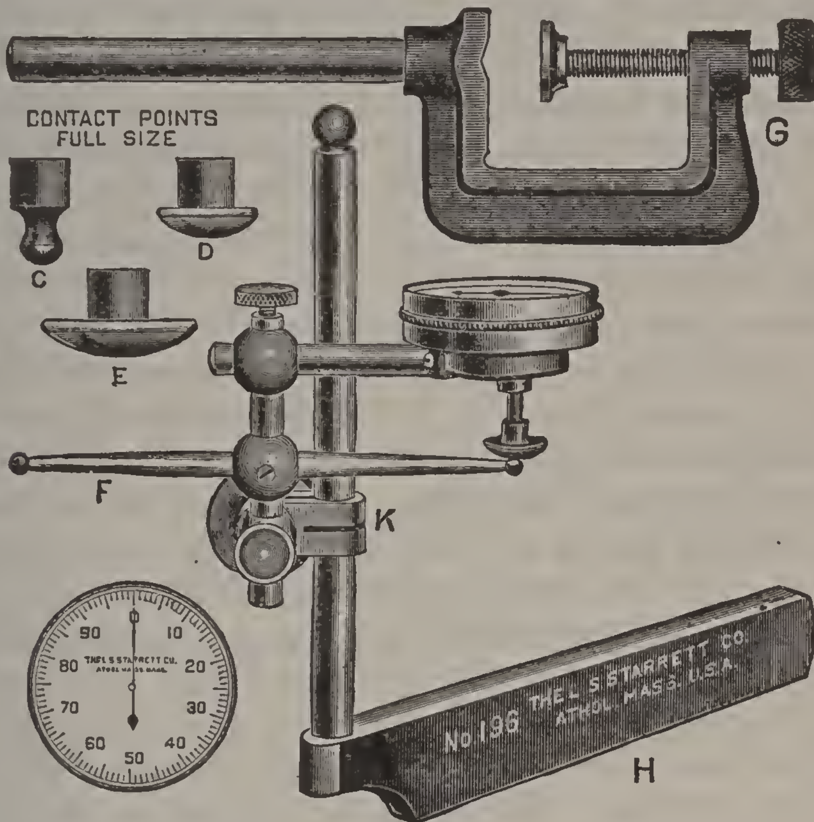


FIG. 102—UNIVERSAL DIAL TEST INDICATOR

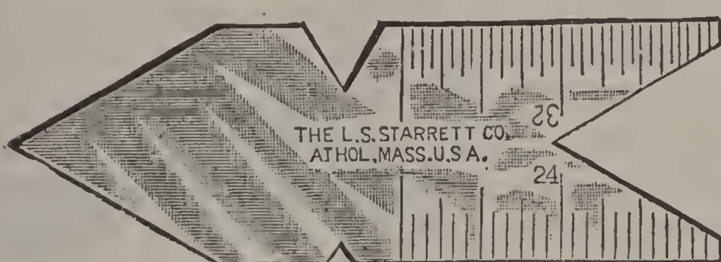


FIG. 103—CENTER GAGE

The cone-points of the centers should be smooth and an exact 60°. These centers should align with each other in the vertical and horizontal planes, and the "live" or head-stock cone-point should rotate truly concentric with its axis.

The trial and error method of adjusting the centers in

alignment is to first bring the cone-points nearly into contact, and by adjusting the tailstock frame upon its cricket bring them into as exact truth as is reasonably possible. With the footstock clamped in position to receive the work, surface the diameter of a trial piece for a length sufficient to allow testing its diameter at several places. If the diameter increases or decreases as the tool passes along the length of the work, readjust the tailstock and repeat the test until the required degree of accuracy is obtained. To test the live center for concentricity, place in the tool-post a universal test indicator, as shown in Fig. 102, with the feeler in touch with the cone-point. Rotate the headstock spindle slowly by hand and note the dial. If the dial shows an eccentricity in excess of the allowed limits for the job to be done, the cone-point should be machined true. In cases where it is customary to have the live as well as the dead center hardened, the cone-point must be trued by some grinding attachment, as, for example, a tool-post grinding fixture. By many workmen the live center is left unhardened and can be trued with a square nose cutting tool and the compound rest—if such is a part of the lathe's construction—and afterward lightly filed to a smooth surface. To test either center for its cone-point angle, use is made of a center gage, shown in Fig. 103.

A common practice to facilitate correct positioning of the live center, which often has to be removed, is to mark in some way both the center and collet, or spindle holding it, so when putting the center back in the spindle the two marks will be in alignment or opposite. This insures a truer center and whenever necessary to grind or turn true the marks should be opposite.

Another common practice, which should be borne in mind and may be well to mention here, is when removing the live center to do work in the chuck, which screws onto the spindle, a piece of waste or even cloth or paper should be stuffed in the hole in the spindle to prevent the collection of oil and chips. If this is neglected, the chips will become imbedded

and prevent the center and collets from running true. Briefly, oil and dirt should never be allowed to gather at either of the centers' bearings.

TEST INDICATOR

This is a tool for indicating minute contact variations upon a graduated dial or upon a graduated arc. The graduations are usually one hundred in a complete circle with an easily read width of spacing. The instrument is built in such a way that one of these spaces represents a movement of the contact-point of 1/1000 inch. Skill and experience on the part of the workman will permit variation of as little as .0005 inch to be read with ease and accuracy. However, if the work is true, there will be no variation or movement of the needle-point.

Various mechanisms are employed for multiplying the movement of the contact-point, all of which are based upon a combination of short and long arm levers.

The test indicator may be used with advantage in any of the common machine tools to indicate eccentricity in the lathe, milling machine, or grinding machine; to indicate uniformity of height in the planer, shaper, boring machine, or milling machine; to indicate parallelism, and to test for alignment in any machine.

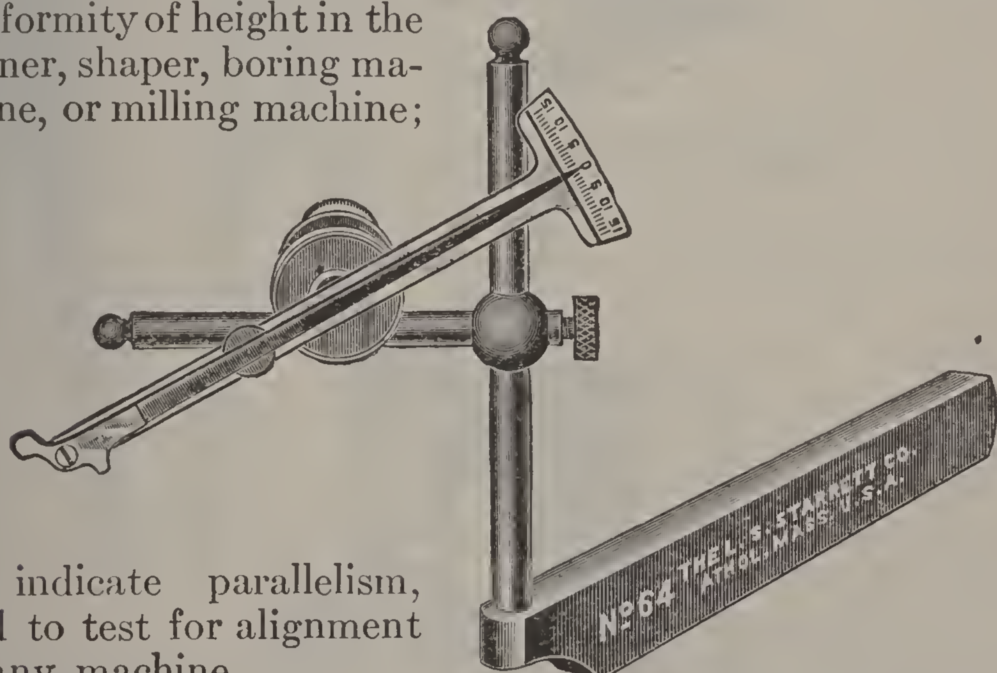


FIG. 104—TEST INDICATOR

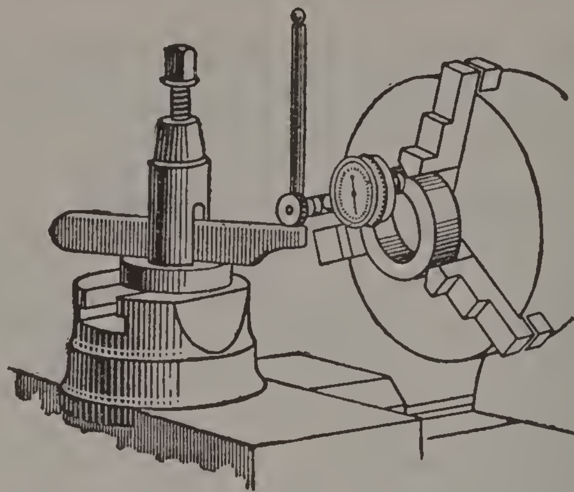


Fig. 105—Truing Work in Chuck

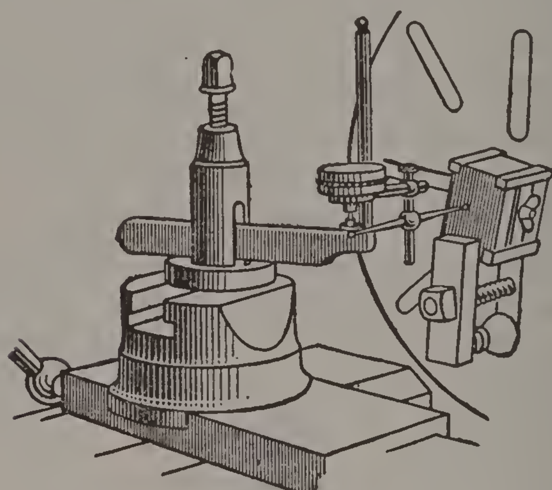


Fig. 106—Truing Jig on Face Plate

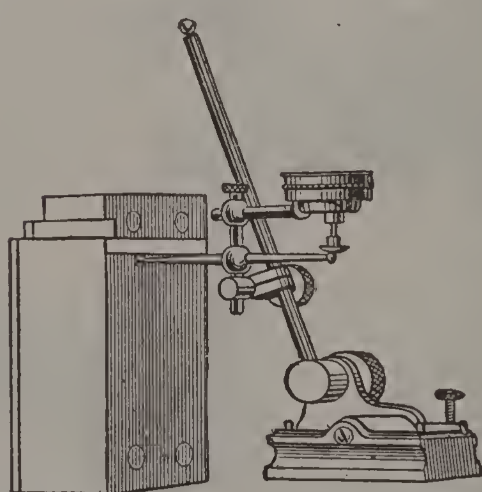


Fig. 107—Indicator Used with Surface Gage on Bench Plate

WORK CENTERS

Most turned work is done upon the lathe centers, and it is necessary to provide suitable cavities in the work, coned to fit the cone-points of the lathe centers. This is termed "centering the work", and consists in first locating the position of the cavities and afterward drilling and reaming them to form and size. The usual practice is to use a combination drill and countersink, as it insures exact concentricity in the hole, as well as being a quick method of doing the work.

LOCATING THE CENTERS

It is evident that the centers should be so located that the entire diameter of the turned job shall finish to size. Beside this, efficient turning demands that the chip taken shall be of practically uniform depth as the work rotates against the cutting tool. For these reasons accuracy in centering is necessary. Where the turned job is made from ordinary bar stock, the centers may be located by scribing lines at an angle across the ends, using a combination square blade with a center head and the scriber provided. In place of this tool, a hermaphrodite caliper may be used to scribe the ends of the stock. The center is located with a center-punch at the intersection of the scribed lines and the concentricity tested by spinning the bar upon the lathe centers. If necessary, the center-punch marks are shifted. If the piece is bent

FIG. 108—HERMAPHRODITE CALIPERS

it must, after centering, be straightened to reasonable truth.

The work should never be straightened in the lathe centers unless it is extremely light. Any strain of bending will throw the lathe centers out of line as well as imposing strains on the lathe parts that they are not intended to receive.

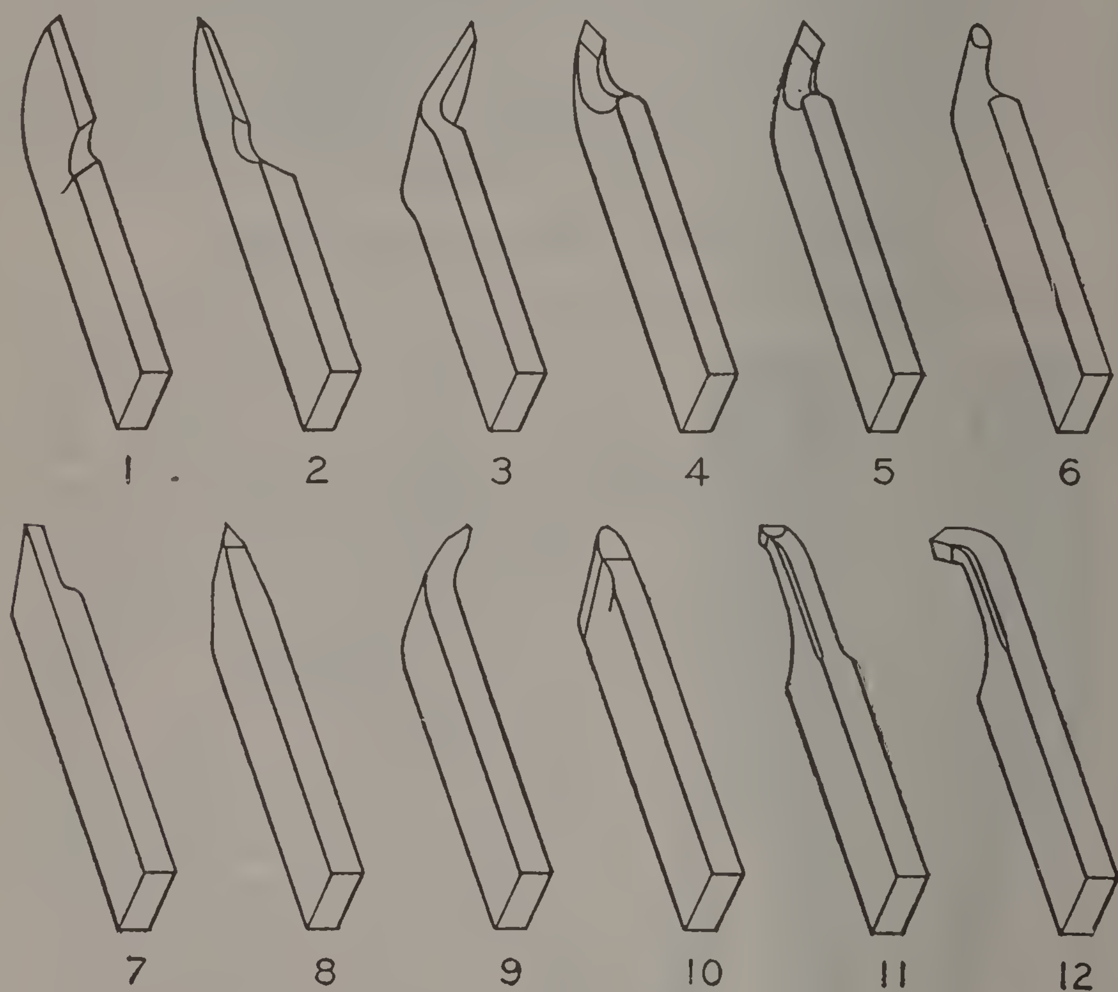


Fig. 109—Typical Lathe Tools

When the job is to be turned from a forging, it is usual to roll the forging on straight edges and scribe lines across the ends, using a surface or height gage. In such cases the forging is so located with reference to the straight edges as to give a fair average of the surface errors due to forging. It is also usual to leave a greater excess of stock for finishing purposes upon a forging than upon rolled bar stock. When the centers

are well located the holes may be drilled under a drill press or in a hand lathe, as convenient. Where much bar stock must be centered a special self-locating centering machine is often used.

LATHE TOOLS

A typical set of tools for use in the engine lathe is shown in Fig. 109, and their uses indicated in Fig. 110A. While in common shop language all these are known as cutting tools, technically speaking, many of them separate the stock in a manner that is analogous to crowding off the metal rather than by a strictly cutting action. Cutting in its proper sense is a splitting action, and a properly ground and properly set cutting tool is a wedge in that it splits off the excess stock. Among the common lathe tools, the side tool (Nos. 1 and 2, Fig. 109), and the diamond-point tool (Nos. 4 and 5, Fig. 109), are the best examples of wedge or splitting action.

The nose of a cutting tool has several sides, two of which come together at some angle to form a cutting edge. The angle formed by these surfaces must be sufficient for strength, and must contain enough metal to conduct away the heat generated by the cutting action. For turning ordinary soft steel and soft gray iron an angle of sixty degrees is good practice. For harder materials the angle may be increased. In the case of forged lathe tools, the working end of the tool is forged upon the end of a short piece of square or rectangular bar stock. The length and size of the shank of the forged tool depend upon the size of chip and the machine used.

Rake

The angle which the upper side of the tool makes with the horizontal is termed the rake. If the slant is away from the work it is termed front rake; if in the direction of the axis of the work, it is termed side rake. A cutting tool may have its upper face forged and ground with either a front or a side rake or a combination of both. (See Fig. 110.)

Clearance

By clearance is meant the angle which the underside of the tool makes with the vertical. (See Fig. 110.) As in the case of "rake", the clearance directly away from the axis of the work or lathe is termed front clearance, while that along the axis of the work is known as side clearance. With the tool in cutting position the clearances must be in any

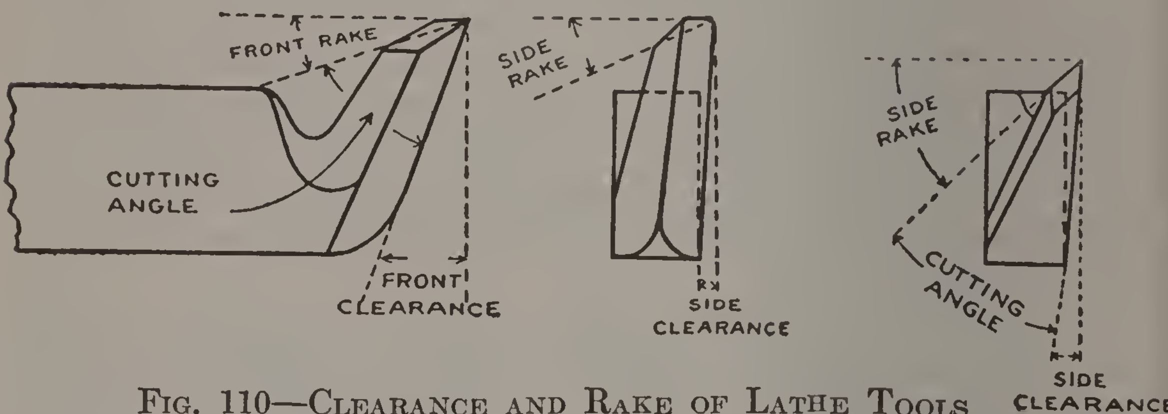


FIG. 110—CLEARANCE AND RAKE OF LATHE TOOLS

case not less than three degrees, and in most cases not more than ten degrees. Experience is the best teacher.

Right-Hand Tools

These are tools having the rake, clearance, and cutting edges formed to turn or square from the right toward the left.

Left-Hand Tools

When the rake, clearances, and cutting edges are formed to cut from the left to the right the tool is known as a left-hand tool.

USES OF LATHE TOOLS

The common uses of standard lathe tools are shown in Fig. 110A. The arrow in the diagram indicates the direc-

tion of tool feed, the work turning toward the tool in all cases.

Left and right hand side tools, Nos. 1 and 2, Fig 110A, are shown in use for facing the side and end of a collar and shaft respectively. Either tool can be fed in or out and can be worked into a shoulder. In some cases a bent or offset side tool, such as No. 3, Fig. 110A, is preferable as it will reach work surrounded by a flange or collar, as for instance a boss or lug on the inside of a brake drum, or wherever a cross-rest would interfere.

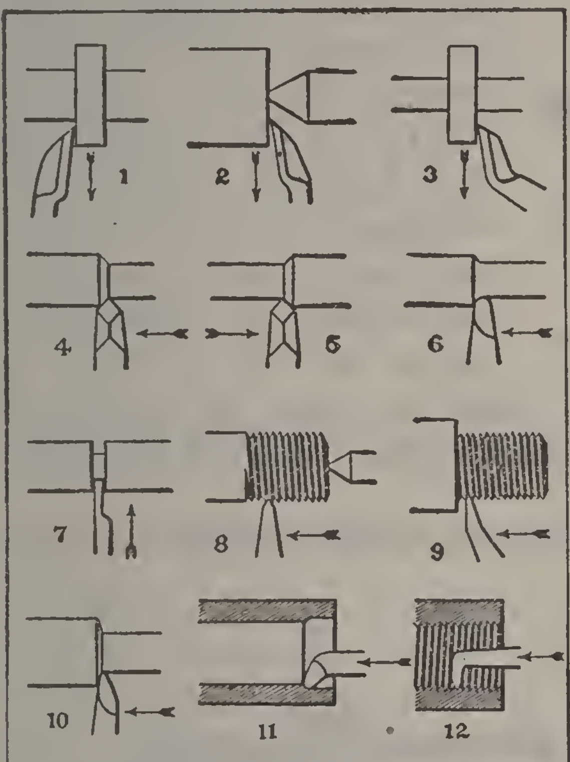


Fig. 110A—Uses of Lathe Tools

For straight turning, the diamond point tools, Nos. 4 and 5, Fig. 110A, are used. The only difference in these two tools is that the rake and clearance are ground on opposite sides so that one tool cuts while feeding to the left, while the other cuts when fed to the right. A round-nose tool, No. 6, Fig. 110A, is very often used for straight turning operations and, like the diamond point tools, may be ground to form either a right or left hand tool.

No. 7, Fig. 110A, is a cutting-off or parting tool and is used for necking down or cutting off stock. In motor work it is often used for relieving shoulders and corners previous to grinding operations.

Three forms of right-hand threading tools are shown in Nos. 8, 9 and 12. Fig. 110A, Nos. 8 and 9, are cutting external and No. 12, internal thread. No. 8 is used for work where the thread does not run up to a shoulder, No. 9 being used

in the latter case. No. 10 is used for rough turning operations and is similar to No. 6, except that it has a longer cutting edge, enabling it to be fed in for a deeper cut.

No. 11 is a boring tool and is ordinarily ground with a round nose except when it is necessary to work into a square shoulder. For such work the boring tool may be ground with a diamond point. Boring tools are often built up from a combination of a tool-holder carrying a cutter bar with the cutting bit inserted as shown in Fig. 110B. A similar type of cutter is commonly used for inside thread cutting, except for very small holes.

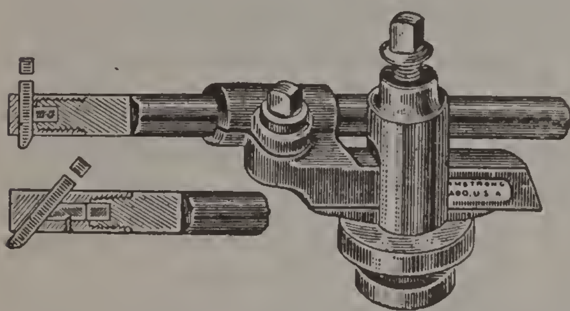


Fig. 110B—Boring Tools with Inserted Cutters

The upper section of the bar shows it fitted with a cutter for straight boring, which may be replaced with a thread cutting tool. The lower section shows the type of cutter used for working close to a shoulder. Fig. 110C shows, mounted in a special tool-holder, a type of boring tool used for boring very small holes.

Fig. 110C—Boring Tool and Holder for Small Diameter Holes

SETTING LATHE TOOLS

It is very important that the lathe tool be properly set in relation to the axis of the work and the direction of the cut. While there are exceptions, notably that of the diamond point, lathe tools are usually set with the cutting point at the exact height of the axis of the lathe. (See Fig. 110D). In the case of the diamond point, the front clearance is

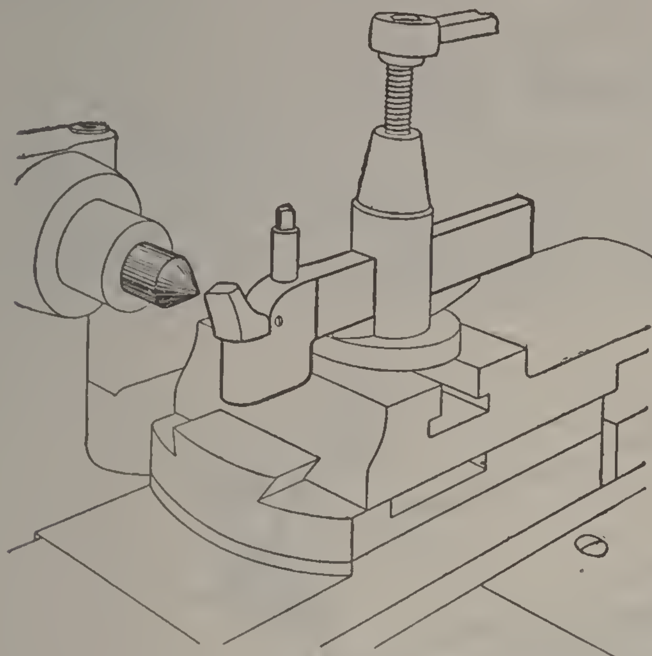


FIG. 110D—THREAD CUTTING TOOL SET AT HEIGHT OF LATHE CENTER

usually forged to fifteen degrees or over. It is necessary, therefore, to set the point above the axis height to obtain a working clearance of not to exceed ten degrees. Five degrees is usually considered correct. (See Fig. 110E.) Unless the cutting tool has a bent shank it is usually set at right angles to the surface of the work.

GRINDING LATHE TOOLS

Lathe tools made from carbon tool steel should be sharpened by grinding upon a *wet* emery-grinder, or upon an ordinary water-drip grindstone. If made from the newer high speed steels, the grinding should be upon a *dry* and rather coarse abrasive wheel. The grinder should have a suitable work-rest upon which to support the tool in sharpening the larger tools, or for resting the hands in the case of the smaller tools.

For purposes of safety, the work-rest should be firmly and securely clamped as close as possible to the used face of the wheel. The grinding may be done upon the periphery of a disk-wheel or upon the sides of a cup-wheel, as desired. In

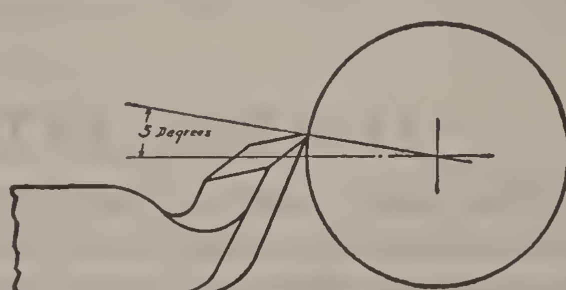


FIG. 110E—CORRECT POSITION OF DIAMOND POINT TURNING TOOL WHEN CUTTING

any case the wheel should rotate to force the tool upon the rest rather than from it, and should run true and in balance. Efficient cutting depends very largely upon the correct sharpening, as well as the correct setting of the cutting tool, and great care should be taken when grinding a lathe tool to have the several faces true and making correct angles with

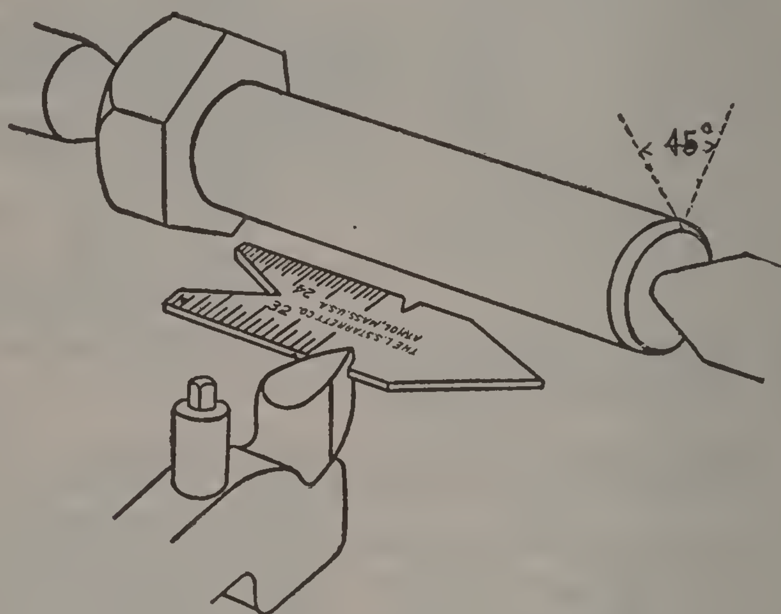


FIG. 111—TESTING THE ANGLE

each other. The manner in which this is done is a pretty good index of the workman's skill. It is good practice to dress the cutting edge of a lathe tool by hand, using a small oil stone, after the tool is ground.

TESTING CUTTING ANGLES

The usual lathe-cutting tools have well-defined cutting edges, and the angularity of the surfaces which meet to form the cutting edge can be measured with a bevel protractor, or, in the case of a sixty-degree angle, the center gage may be used. The center gage is also used to test the angle when grinding a vee-pointed thread tool, as illustrated in Fig. 111.

TOOL HOLDERS

The high cost of the materials used for modern cutting tools has resulted in the marketing of a variety of holders designed to hold cutting points and by their use a large number of relatively inexpensive cutting points are made to

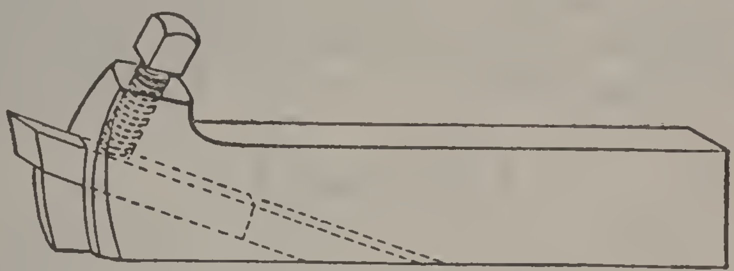


Fig. 112—Tool Holder

interchange in a single shank or holder. One form of tool holder is made to hold all points forged in the regular forms shown in Fig. 109. In some cases, however, the holders are made to carry short bits broken from square bar stock and which are afterward sharpened into some resemblance to the true forged shape. (See Fig. 112.)

MATERIALS FOR CUTTING TOOLS

These are known as carbon steel (tool steel), high speed steel, and a product of the electric furnace sold under the trade name of "Stellite". Carbon steel, or, as it was formerly termed, "tool steel", is high in carbon, eighty point to one hundred and twenty-five point, and when correctly heated and afterward plunged in cold water, hardens to a very high degree. Unfortunately for high speed cutting, the hardness is drawn at a comparatively low heat, and care must be taken not to overheat or blue it.

High speed steel is a special steel having its composition alloyed with tungsten and sometimes vanadium or molybdenum. While heat treatment does not give it the exceed-

ing hardness of tool or carbon steel, high speed steel has the peculiar property of retaining its hardness at temperatures considerably in excess of those which readily soften tool steel. Tools made from high speed steel are used at speeds, feeds, and cuts which heat the tools and chips to a dull red.

Stellite is a cutting material containing chromium, cobalt, and sometimes tungsten. It is cast into form and cannot be forged. Its hardness is practically equal to the diamond, and under favorable conditions marvelous turning may be done.

MANDRELS

Where the work is to be turned true with a hole through it, as, for example, gear blanks, etc., work centers must be provided for holding it on the lathe centers. The common way is to force or drive into the work-hold a bar having center holes in its ends. This bar should be classed as a tool-room tool, and is properly known as a mandrel, although often called an arbor.

A standard set of mandrels varies in diameter and in length, according to the shop conditions. They are made of either tool steel hardened and ground true with the centers, or from machinery steel, pack hardened and afterward ground. The ends for a short distance are reduced in diameter and provided with flats for clamping on the dog. Mandrels usually taper at the rate of .0005″ in an inch. The diameter of the hole fitted by the mandrel is stamped upon the larger end. As the quality of the work depends upon the truth of the mandrel it should be tested upon dead centers with a test indicator before being used. To use, drive or force it into place, using a mandrel press for forcing or a lead hammer for driving, carefully removing dirt, chips, or pieces of lead from the centers before placing the work in a lathe. Lathe drive with the usual lathe-dog, as for any job done on the centers. Avoid forcing or driving the mandrel into a hole that is neither round nor straight. Also avoid scoring the mandrel with the cutting tool.

THREAD CUTTING ON A LATHE

When screw threads are cut in an engine lathe, the point of the cutting tool is shaped to the exact form of the spaces between threads. By means of a lead screw and a train of gearing the tool is compelled to move along the axis of the work at a definite rate of advance as the work rotates. As the train of gears usually furnished with an engine lathe can be changed to give different rates of advance, it is in this manner possible to cut threads of a large variety of pitches. In practice, a set of several gears having different numbers of teeth are furnished with each lathe. Those furnished will usually provide for cutting all the threads within the range of the lathe with which they come. These are known as "change gears" and their use is obvious.

Among the tools listed in Fig. 109 were shown ordinary threading tool points (Nos. 8, 9 and 12). The points shown have sides at an angle with each other of 60 degrees, but it is obvious that these or any other form of point must be formed and tested to give the correct form of thread desired in any particular case.

Use lard oil when threading steel, wrought and malleable iron, and use plenty of lubricant. Cut the cast metals dry.

Selecting Change Gears*

Given—or having ascertained by use of a thread pitch gage—the number of threads per linear inch to be cut, and the number of threads per linear inch of the lead screw, the problem is to select gears giving the desired ratio of cut to lead screw. For example, it is desired that seven single threads per linear inch shall be cut upon a 1¼ inch bolt, and it is found by scaling that the lathe lead screw had five single threads per linear inch. The ratio of cut to lead screw is then that of seven to five (7/5). The change gears selected should, therefore, be as seven is to five. If both members of a fraction are multiplied by the same number, the ratio is

* See also pp. 162-168, Vol. I, Starrett Books.

not changed. This allows of raising the fraction to suit the gears which are in the set furnished; for example

$$\frac{7}{5} \times \frac{5}{5} = \frac{35}{25}$$

If gears having thirty-five teeth and twenty-five teeth, respectively, are found in the furnished set, the selection of these gears will give, when rightly placed, the desired tool advance for cutting seven threads per linear inch.

The directions above refer to the most simple form of lathe. Various lathe manufacturers have introduced different arrangements of the gearing, and most lathes today have some form of quick change gear box, but with any lathe the above procedure will give correct results if it is first determined what number of threads per inch will be cut if gears of the same number of teeth are placed on spindle stud and lead screw. This number called the "lathe screw constant" should then be considered as being the number of threads on the lead screw even though it is not the actual number. Most lathe manufacturers provide a table, attached to the lathe, which indicates at a glance the proper selection of gears for cutting threads of varying pitches.

Placing the Change Gears

The common engine lathe—if not equipped with the quick change gear box—has projecting through its headstock a shaft known as the "stud". This projects a sufficient distance to allow of mounting gearing and usually the upper cone for the feed belt. Gears mounted or to be mounted upon this projecting stud are termed "stud gears". Those mounted upon the projecting end of the lead screw are known as lead gears. When the number of threads to be cut is more per linear inch than that of the lead screw, the smaller of the selected gears is placed upon the "STUD" and the larger upon the lead screw. In the example given under

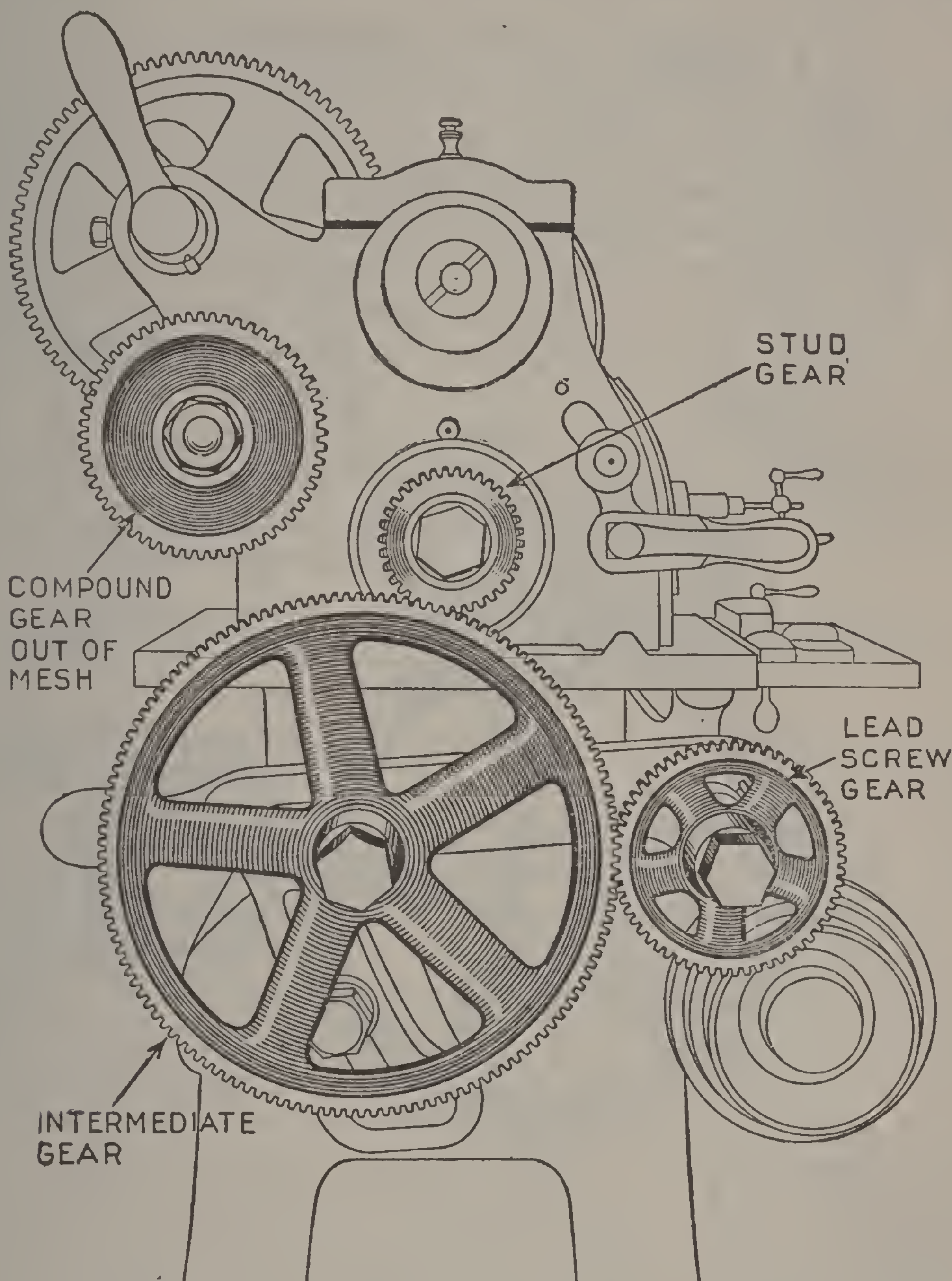


FIG. 113—SIMPLE TRAIN OF GEARS FOR THREAD CUTTING

"Selecting Change Gears", assuming that the stud rotates in unison with the lathe spindle, the 25-tooth gear would be placed on the stud and the 35-tooth gear on the lead screw. Reverse the order if the number of threads per linear inch

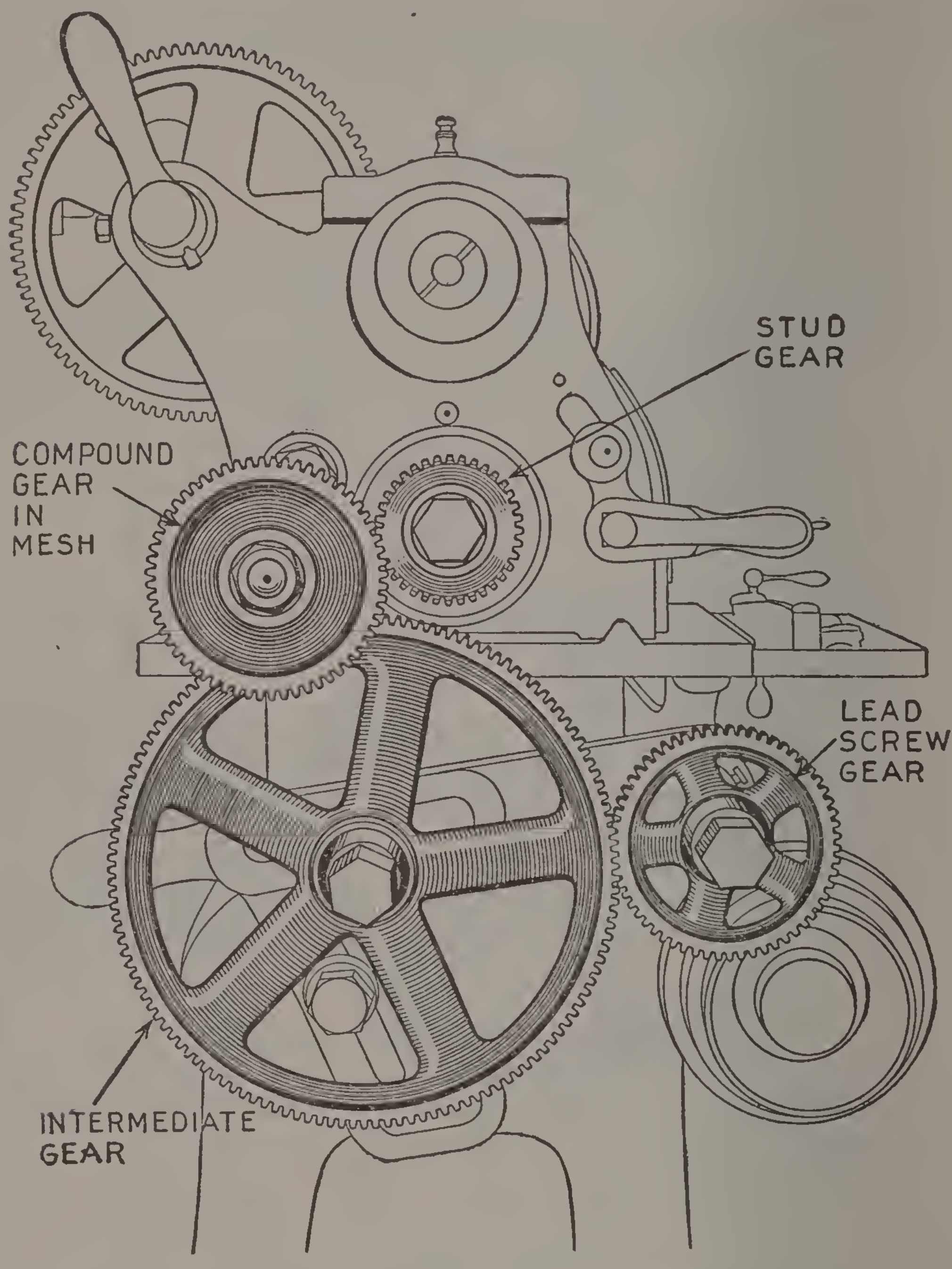


FIG. 114—COMPOUND GEARS FOR THREAD CUTTING

is less than that of the lead screw. The number of teeth in the large idler gear has no bearing upon the results, as it simply conveys the motion of the upper or stud gear to the lower or lead-screw gear.

Compounding Gears

As a means of enlarging the range of threads per linear inch possible to be cut with any set of change gears, most lathes are provided with an adjustable compound auxiliary stud which is provided with two locked gears having a ratio each to the other of two to one. As an example of their use, assume that a gear having ninety teeth was needed upon the lead screw to cut a given number of threads. If the set of gears furnished failed to provide a ninety gear, but did provide one of forty-five teeth, placing this on the lead screw and meshing the two to one compound stud into the train completes the desired ratio, and advances the tool as if a 90-tooth gear had been used.

MEASURING AND TESTING SCREW THREADS

For ordinary purposes, screw threads when cut are fitted to some threaded hole. This may be a hardened and ground gage, or may be an ordinary threaded nut, depending upon the accuracy of the work. Where the quality of the work demands special accuracy, or where standard thread gages are not available, the thread is tested by measurements made with calipers. If the point of the thread tool has been carefully and exactly formed and accurately set in place, measuring the diameter at the root of the thread may give sufficiently accurate results, and this may be done with a set of thin point spring calipers. (See Fig. 115.) When greater accuracy is required, micrometers having special thread-measuring points are resorted to. (See Fig. 116.) In all this it is assumed that the thread tool is ground, set, and operated to a given exact thread outline.

MEASURING LATHE WORK

Work done in the engine lathe is of such a variety that a considerable list of measuring tools may be needed to cover all cases. Ordinarily, however, the diameter measurements

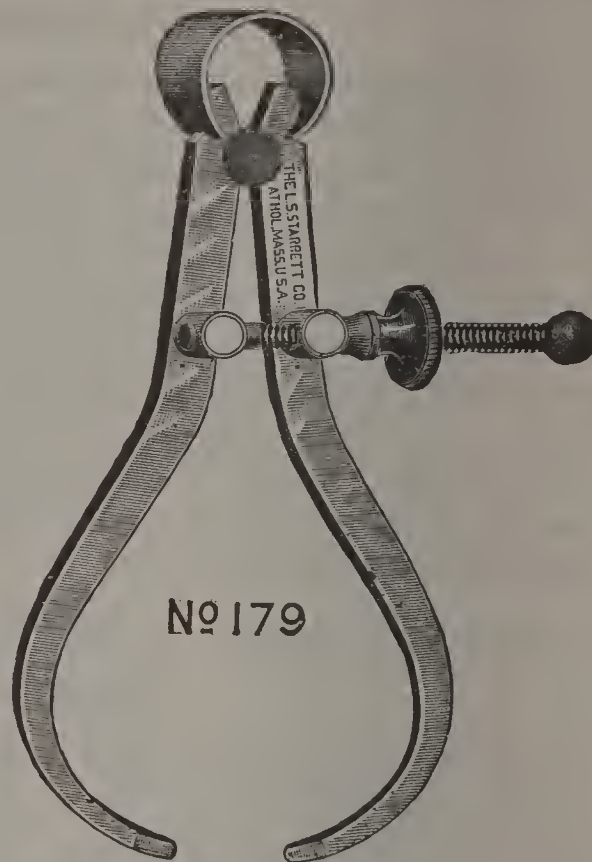


Fig. 115—Outside Thread Calipers

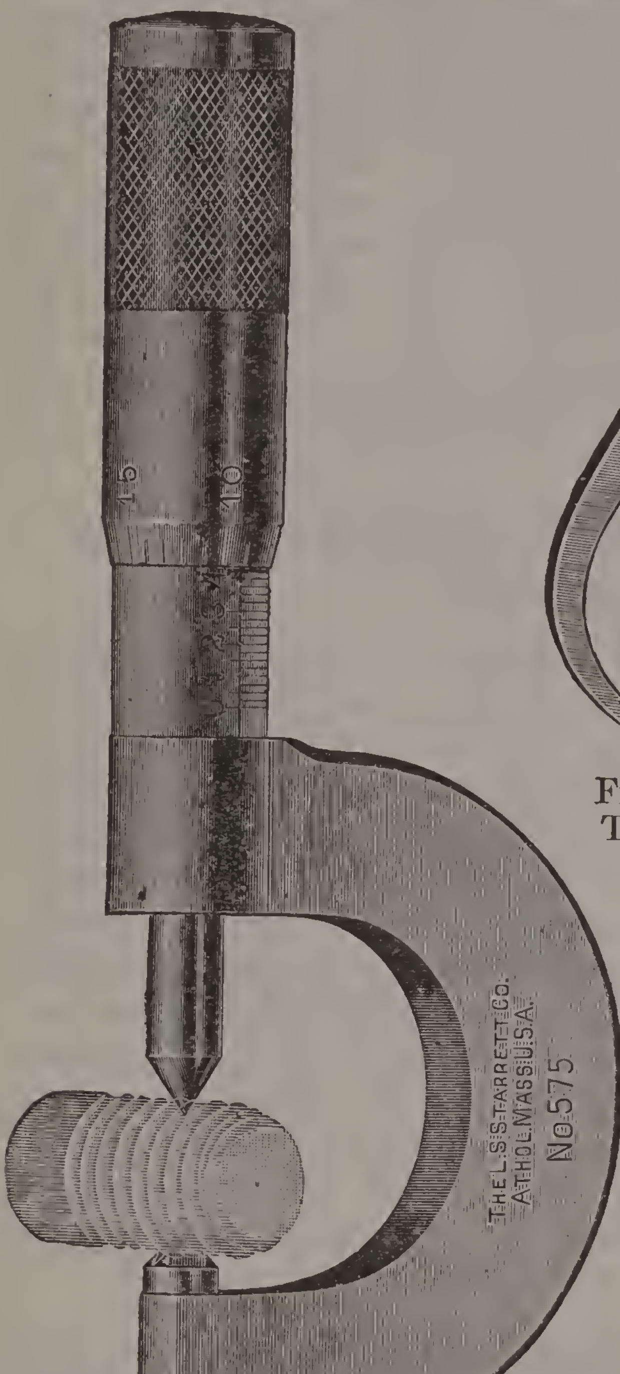


Fig. 116—Thread Micrometer Caliper

can be made with spring calipers, micrometers, or some of the usual bar calipers. Cylindrical plug and ring gages, as well as limit snap gages, are sometimes used for diameter measurements—especially on production work—and many of these may be used in measuring the shorter lengths. For the longer measurements of length, caliper squares are provided. The more accurate measurements are usually made by using a micrometer.

TAPER TURNING*

Where two parts are to fit firmly together when in use, as, for example, centers into lathe spindles, automobile shafts, etc., and it is desirable to have them easily removable, what are known as taper fits are used. For this purpose several rates of change in diameter have become standards. The B. & S. Standard is in general use for the spindle tapers in milling machines. The Morse taper is the one commonly used for all drills and drilling machinery. Either of these may be used for the tapered hole in lathe spindles, while some lathe manufacturers have established standards of their own.

Ordinary tapers are rated at the amount which the diameter changes in a foot's length; as, for example, the Brown & Sharpe taper of ½ inch per foot. To turn a taper it is necessary to use a lathe provided with a taper attachment or to adjust the footstock of the engine lathe sufficiently off center to give the required

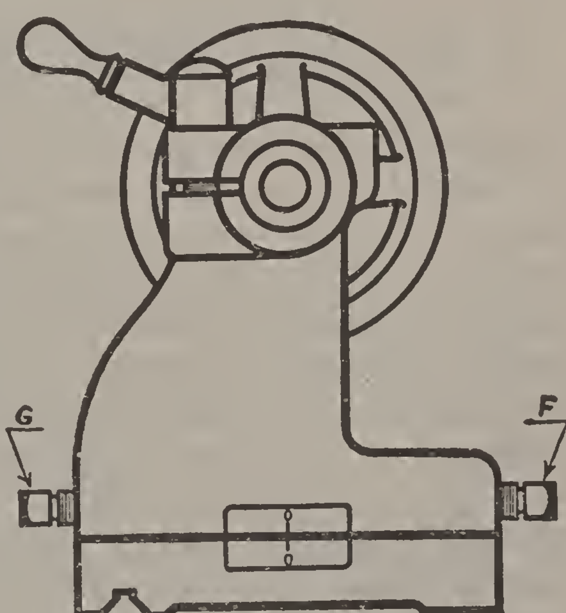


Fig. 117—Lathe Tailstock with Adjusting Screws

* See also pp. 86-91, Vol. I, Starrett Books, and pp. 114-121, Vol. II, Starrett Books.

rate of diameter change. As all taper attachments are graduated to read direct, they are easily set for the required taper. Adjustment of the tailstock of an engine lathe to turn a taper is not so simple as to use the taper attachment.

TAPER TURNING WITHOUT TAPER ATTACHMENT

If the distance the center points enter the work or the mandrel is ignored, the mandrel length can be considered as the distance apart of the center points. The calculation necessary to determine the distance which the centers shall be offset, is that of multiplying the length of the work or mandrel in feet by one-half of the required taper in inches. To turn a Brown & Sharpe taper on a piece of work 9 inches long the problem would work out as follows:

$$\frac{.500}{2} \times \frac{9}{12} = 0.1875 = \frac{3}{16}$$

and the footstock would be set over 3/16 inch.

In the above illustrative example both length and amount of taper are given, but the amount of taper is not always known. Suppose a piece is 8 inches long and a taper is to be turned on one end, the tapered portion to be 4 inches long. The difference in diameters of these 4 inches is to be ½ inch. How much must the tailstock be offset? If the taper is ½ inch in 4 inches it would be 1½ inches in a foot and the tailstock would be moved over one-half of 1½ inches or ¾ inch, if the piece were a foot long, but as it is only 8 inches or 2/3 of a foot long, the tailstock should be moved over 2/3 multiplied by ¾ or ½ inch. Had the piece been 18 inches long, it would have been necessary to move the tailstock over 1½ (or 3/2) multiplied by ¾ or 1⅛ inches.

It has been assumed for these simple calculations that the lathe centers merely touch the ends of the piece, thus making the length of the piece the same as the distance between centers. But in actual work the distance the centers enter

the piece must be considered. The calculation should be as accurate as possible to avoid continually changing the tailstock to get a reasonably good taper fit. The necessity of considering the distance the center enters the piece depends somewhat upon its length. If the piece is very long, the actual taper will differ considerably from the calculated taper. If each center enters the piece one-fourth inch they would enter a total of one-half inch, and the length of the piece should be reduced by one-half inch in the calculation. While turning the taper the calipers should be used frequently so that it may be soon determined whether or not the tailstock is correctly placed.

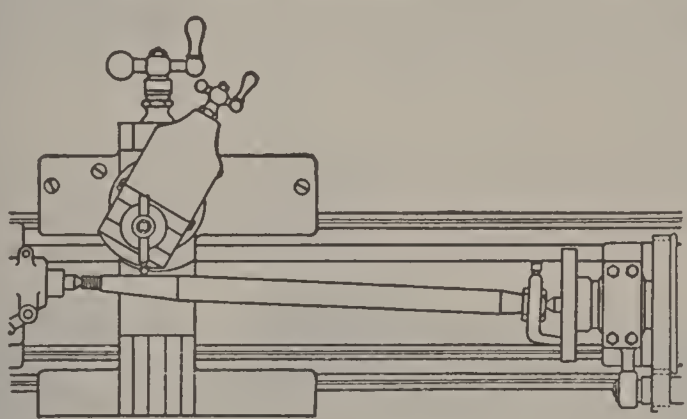


FIG. 118—LATHE SET-UP FOR TURNING TAPER WITHOUT TAPER ATTACHMENT

Lathes adapted to taper turning without the use of a taper attachment have tailstocks which can be shifted off the dead-center line of the lathe spindle. On such tailstocks will be found two indicating lines marked zero, as shown in Fig. 117. If these markings are exactly aligned, the live spindle center of the lathe is in line with the tailstock center.

To set the tailstock off center, it is only necessary to loosen one of the two set screws (F and G, Fig. 117) and screw in the other the same amount, or until it strikes the tailstock hub. Then clamp the tailstock to the bed and set up the work so that it occupies the position as shown in Fig. 118.

The amount the tailstock shall be set off-center can be

determined only by the judgment of the machinist and will, of course, vary with the length of the taper, making it necessary to test the work by taking a light chip from the small end of the work and trying it in the taper socket to which the part is to be fitted.

To test a taper, as in the case of an automobile shaft, it should be pressed lightly into a standard tapered hole and worked back and forth sufficiently to mark the places where bearing occurs. If the work has been lightly covered with some marking pigment, or with chalk, the bearing points will be more distinct. Care, however, must be taken that the coating is not sufficient to smooch, as it will deceive the workman. Adjust taper-setting until a correct fit is obtained.

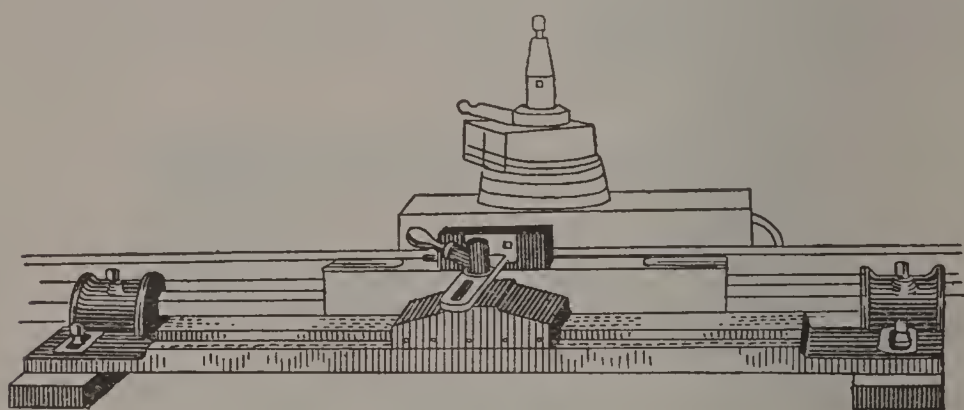


Fig. 119—Taper Turning Attachment on Lathe

The use of a taper turning attachment, such as is shown in Fig. 119, greatly simplifies the work. The taper attachment illustrated, is fitted to the rear V of the lathe by two clamps, one at either end. A slide on the attachment regulates the degree of taper. The attachment is connected by an arm to the tool slide, which is then disengaged from its feed-screw. As the lathe carriage moves along the bed, the tool slide is moved in or out, according to how the slide of the taper attachment is set, thus guiding the cutting tool and producing the proper taper. Where the taper attachment is used, all that is required of the operator is to measure

AMOUNT OF TAPER IN A GIVEN LENGTH

(Expressed in 1000ths of an Inch)

Length of Taper	Taper in Inches per Foot										
	1/16	3/32	1/8	1/4	3/8	1/2	0.600	5/8	3/4	1	1 1/4
1/32	.1628	.2441	.3255	.6510	.9766	1.302	1.563	1.628	1.953	2.604	3.255
1/16	.3255	.4883	.6510	1.302	1.953	2.604	3.125	3.255	3.906	5.208	6.510
1/8	.6510	.9766	1.302	2.604	3.906	5.208	6.250	6.510	7.813	10.42	13.02
3/16	.9766	1.465	1.953	3.906	5.859	7.813	9.375	9.766	11.72	15.63	19.53
1/4	1.302	1.953	2.604	5.208	7.813	10.42	12.50	13.02	15.63	20.83	26.04
5/16	1.628	2.441	3.255	6.510	9.766	13.02	15.63	16.28	19.53	26.04	32.55
3/8	1.953	2.930	3.906	7.813	11.72	15.63	18.75	19.53	23.44	31.25	39.06
7/16	2.279	3.418	4.557	9.115	13.67	18.23	21.88	22.79	27.34	36.46	45.57
1/2	2.604	3.906	5.208	10.42	15.63	20.83	25.00	26.04	31.25	41.67	52.08
9/16	2.930	4.395	5.859	11.72	17.58	23.44	28.13	29.30	35.16	46.88	58.59
5/8	3.255	4.883	6.510	13.02	19.53	26.04	31.25	32.55	39.06	52.08	65.10
11/16	3.581	5.371	7.161	14.32	21.48	28.65	34.39	35.81	42.97	57.29	71.61
3/4	3.906	5.859	7.813	15.63	23.44	31.25	37.50	39.06	46.88	62.50	78.13
13/16	4.232	6.348	8.464	16.93	25.39	33.85	40.63	42.32	50.78	67.71	84.64
7/8	4.557	6.836	9.115	18.23	27.34	36.46	43.75	45.57	54.69	72.92	91.15
15/16	4.883	7.324	9.766	19.53	29.30	39.06	46.88	48.83	58.59	78.13	97.66
1	5.208	7.813	10.42	20.83	31.25	41.67	50.00	52.08	62.50	83.33	104.2
2	10.42	15.63	20.83	41.67	62.50	83.33	100.0	104.2	125.0	166.7	208.3
3	15.63	23.44	31.25	62.50	93.75	125.0	150.0	156.3	187.5	250.0	312.5
4	20.83	31.25	41.67	83.33	125.0	166.7	200.0	208.3	250.0	333.3	416.7
5	26.04	39.06	52.08	104.2	156.3	208.3	250.0	260.4	312.5	416.7	520.8
6	31.25	46.88	62.50	125.0	187.5	250.0	300.0	312.5	375.0	500.0	625.0
7	36.46	54.69	72.92	145.8	218.8	291.7	350.0	364.6	437.5	583.3	729.2
8	41.67	62.50	83.33	166.7	250.0	333.3	400.0	416.7	500.0	666.7	833.3
9	46.88	70.31	93.75	187.5	281.3	375.0	450.0	468.8	562.5	750.0	937.5
10	52.08	78.13	104.2	208.3	312.5	416.7	500.0	520.8	625.0	833.3	1042
11	57.29	85.94	114.6	229.2	343.8	458.3	550.0	572.9	687.5	916.7	1146
12	62.50	93.75	125.0	250.0	375.0	500.0	600.0	625.0	750.0	1000	1250
13	67.71	101.6	135.4	270.8	406.3	541.7	650.0	677.1	812.5	1083	1354
14	72.92	109.4	145.8	291.7	437.5	583.3	700.0	729.2	875.0	1167	1458
15	78.13	117.2	156.3	312.5	468.8	625.0	750.0	781.3	937.5	1250	1563
16	83.33	125.0	166.7	333.3	500.0	666.7	800.0	833.3	1000	1333	1667
17	88.54	132.8	177.1	354.2	531.3	708.3	850.0	885.4	1063	1417	1771
18	93.75	140.6	187.5	375.0	562.5	750.0	900.0	937.5	1125	1500	1875
19	98.96	148.5	197.9	395.8	593.8	791.7	950.0	989.6	1188	1583	1979
20	104.2	156.3	208.3	416.7	625.0	833.3	1000	1042	1250	1667	2083

Example: For the amount of taper in 33 3/8 inches, when the taper is 5/8 inch per foot, add 1042 (for 20 inches) to 677.1 (for 13 inches) and 19.53 (for 3/8 inch); the result is 1738.63 thousandths, or 1.739 inches.

accurately the taper on the piece he wishes to duplicate or fit and set the slide to correspond. Remember that any taper actually set on the slide is doubled in the turning operation. To illustrate: If in a distance of three inches measured accurately along the surface of the taper to be matched or fitted, it is found that the diameter of the piece or hole increases exactly ½ inch, then the taper attachment slide, measuring from the V-guides of the lathe, must be set to show a taper of ¼ inch in three inches of length. Even when using a taper attachment it is advisable to test the work—as was done when taper turning without the attachment—before the finish cut is taken.

RULES FOR FIGURING TAPERS

To Find	Having Given	Method
Taper per inch.	Taper per foot.	Divide by 12.
Taper per foot.	Taper per inch.	Multiply by 12.
Taper per foot.	End diameters and length of taper, both in inches.	Find difference between diameters; multiply by 12 and then divide by length of taper.
Diameter at small end, in inches.	Diameter at large end, length of taper in inches, and taper per foot.	Multiply taper per foot by length of taper and divide by 12; subtract result from diameter at large end.
Diameter at large end, in inches.	Diameter at small end, length of taper in inches, and taper per foot.	Multiply taper per foot by length of taper and divide by 12; add result to diameter at small end.
Amount of taper in inches.	Taper per foot and given length in inches.	Multiply given length by taper per foot and divide by 12.
Distance in inches between two given diameters.	Taper per foot and the two diameters in inches.	Find difference between diameters; multiply by 12 and then divide by taper per foot.

ECCENTRIC TURNING

While for the most part the lathe is used for work exactly concentric with the axis, it can be used for turning work that is not concentric and which is termed "eccentric". An example of such work is seen in the eccentrics which operate the valves of steam engines, and in crankshafts, etc. If the work has a hole through it, as in the above example, the hole

CUTTING SPEEDS AND FEEDS FOR TURNING TOOLS

High Speed Steel Tools; from Transactions A. S. M. E., Volume 28; F. W. Taylor.

Cutting speed in feet per minute for a tool which is to last 1½ hours without regrinding.

Inches		Cutting Steel		
Depth of Cut	Feed	Soft	Med.	Hard
Standard ¾-inch Tool				
3/32	1/64	465	233	106
	1/32	302	156	70.9
	1/16	209	105	47.6
	3/32	165	82.8	37.2
1/8	1/64	413	207	93.9
	1/32	277	139	62.9
	1/16	186	92.9	42.2
	1/8	123	61.6	28
3/16	1/64	350	175	79.6
	1/32	235	118	53.4
	1/16	157	78.8	35.8
	3/32	125	62.4	28.3
1/4	1/64	313	157	71.2
	1/32	210	105	47.8
	1/16	141	70.5	32
3/8	1/64	269	135	61.3
	1/32	181	90.4	41.1
	1/16	121	60.5	27
Standard 1-inch Tool				
3/32	1/64	490	245	111
	1/32	340	170	77.2
	1/16	235	118	53.5
	3/32	189	94.6	43
1/8	1/64	427	214	97.2
	1/32	296	148	67.3
	1/16	205	102	46.6
	1/8	142	71	32.3
3/16	1/32	247	124	56.2
	1/16	171	85.6	38.9
	1/8	119	59.3	26.9
	3/16	95.4	47.7	21.7
1/4	1/64	314	157	71.4
	1/32	218	109	49.4
	1/16	151	75.3	34.2
	1/8	104	52.1	23.7
3/8	1/64	265	133	60.3
	1/32	183	91.9	41.8
	1/16	127	63.6	28.9
	3/32	102	51.2	23.3
1/2	1/64	234	117	53.2
	1/32	162	80.9	36.8
	1/16	112	55.9	25.4
	3/32	90	45	20.5

Inches		Cutting Cast Iron		
Depth of Cut	Feed	Soft	Med.	Hard
Standard ¾-inch Tool				
3/32	1/32	165	82.5	48.1
	1/16	118	58.9	34.4
	1/8	81.3	40.7	23.7
	3/16	65	32.5	19
1/8	1/32	151	75.5	44
	1/16	108	53.9	31.4
	1/8	74.4	37.2	21.7
	3/16	59.4	29.8	17.4
3/16	1/32	134	67.1	39.1
	1/16	95.7	47.9	27.9
	1/8	66.1	33	19.3
	3/16	52.9	26.4	15.4
1/4	1/32	124	61.9	36.1
	1/16	88.4	44.2	25.8
	1/8	61.1	30.5	17.8
3/8	1/32	111	55.6	32.5
	1/16	79.3	39.6	23.1
	3/32	64	32	18.7
Standard 1-inch Tool				
3/32	1/32	180	90.2	52.7
	1/16	133	66.4	38.7
	1/8	94.5	47.2	27.6
	3/16	76.9	38.5	22.5
1/8	1/32	162	81.2	47.4
	1/16	120	59.8	34.9
	1/8	85.1	42.5	24.8
	3/16	69.3	34.6	20.2
3/16	1/32	141	70.5	41.4
	1/16	104	51.8	30.2
	1/8	73.8	36.9	21.5
	3/16	60.1	30	17.5
1/4	1/32	128	64	37.3
	1/16	94.2	47.1	27.5
	1/8	67.1	33.5	19.6
	3/16	54.6	27.8	15.9
3/8	1/32	112	56.2	32.8
	1/16	82.7	41.4	24.1
	1/8	58.9	29.4	17.2
	3/16	47.9	24	14
1/2	1/32	103	51.4	30
	1/16	75.6	37.8	22.1
	1/8	53.8	26.9	15.7
	3/16	43.8	21.9	12.8

is first finished to required dimensions. A mandrel is then used for carrying the work on the centers. While the mandrel has been built on one set of centers exactly true with its axis, for eccentric turning it has a second set of centers which are offset the amount required for the eccentricity specified. In the case of eccentrics made solid with the shaft, the two sets of centers, one for turning the shaft and the other for finishing the eccentrics, are made side by side in the ends of the shaft, as shown in Fig. 120.

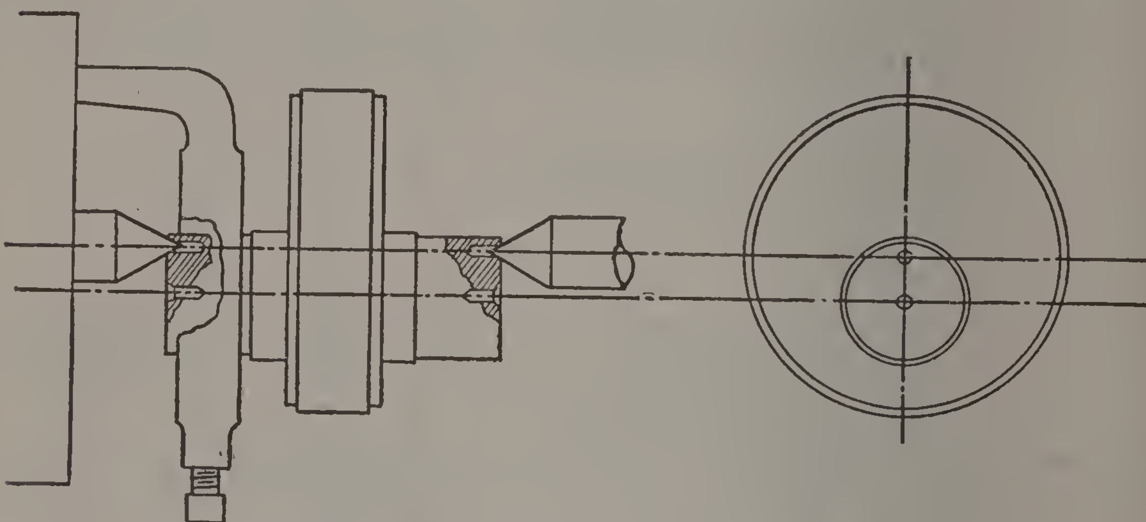


Fig. 120—Finishing Eccentrics

When the specified eccentricity is too extreme to allow both pairs of centers coming within the limits of the diameter of the shaft, special ends may be cast or forged on the ends of the work, and can afterward be machined off. In crank-shaft turning, special attachments are provided for the ends of the shaft. Special eccentric turning chucks also may be made to hold the work.

CHUCKING

Chucking includes not only the mounting of the work in the chuck, but performing the necessary operations on it while so held. The name "chuck" is given to devices having

a variety of forms, all of which are designed to hold work or tools upon or in the nose of a spindle. In general, the heavier sorts are mounted upon a face-plate which screws upon the end of the spindle, while smaller sizes are fitted with a taper shank which fits tightly into the tapered hole in the spindle. These smaller sizes are used for carrying tools such as drills, screws, studs, wire pins, etc., and are known as drilling and spring chucks.

The larger sizes are widely used for holding work for machine operations, and are sometimes called "work-chucks". On their face they are provided with adjusting jaws movable regularly to and from the center; these jaws are so designed that a considerable variety of work may be readily held and successfully worked upon with common cutting tools. If the jaws are moved by means of screws or gears, and can be adjusted independently, the chuck is called an independent jaw-chuck. If all the jaws are made to move together, it is known as a Universal chuck.

Holding the Work

The work must be clamped firmly in the chuck while being machined. Care must also be taken that the clamping of a slender piece is not so firm as to distort or spring it. If work slips, tools may be broken, and if held too tightly and sprung or crushed, the work is injured and in some cases entirely ruined.

Truing the Work

Adjusting the chuck-jaws so that the work will run as true as desired is termed "truing up the work". This is preliminary to any tooling which may be done on the job. Often this truing of the work can be accomplished by holding a piece of chalk to just touch the work, leaving a plain marking. Where greater accuracy is required, the work is indicated with a Universal Dial Test Indicator or Universal Test Indicator.

KNURLING

The surfaces of adjusting screws and small machine parts are often given a regular rough surface for easy gripping. In the machine shop this is done by using a tool known as a "knurl" or "knurling tool", which consists of one or more indented rollers or knurls mounted to rotate in some form of holder.

These knurls are forced into and fed along the stock until the indented design has been sufficiently imprinted into the surface. When neatly and effectively done the results give a fine gripping surface and a rather pleasing effect to the eye. The knurling tool may be fed along the surface of the work by hand, but usually the power traverse feed is used. The process is repeated if one passage of the tool does not give sufficient depth.

LAPPING

Round work that is to run in bearings and bushings such as crankshafts, propeller shafts, etc., must be finished to a smooth, true surface after having been turned to size. For this purpose a lap is used. A lap consists of any kind of device desired that will clamp around the shaft and hold some fine grinding compound while the work revolves. The most simple form of lap is made by hinging two pieces of wood together, cutting a recess for the shaft and filling this in with old pieces of belt leather. A mixture of oil and grinding compound is then put on the leather, the lap clamped on the shaft and the shaft turned until it has the desired polish. Off-center parts such as the crankpins can be done very easily if the lathe is placed in back gear so that the work runs slowly. The lap should move back and forth as the work turns around.

LATHE WORK

So many special purpose machines have been developed for automobile service that the universatility of the lathe is sometimes lost sight of by those who have not learned the machin-

ist's trade. Very often an otherwise impossible boring job can be done on a lathe with a boring bar which is simply a steel bar of sufficient diameter so that it will not spring and is provided with centers and a flat end for the attachment of the lathe-dog. At convenient points there are square holes through the bar to accommodate the self-hardening steel cutters. These are held in place with set screws. By clamping the work done on the lathe carriage in exactly the right position, the boring bar cutters cut a hole the size they are set for, the rod or screw being used for the feed. The cross slide on the lathe is usually removed when using a boring bar.

When truing long thin work it is best to use the steady rest which is part of the lathe equipment. This keeps the work from springing and helps to get it true.

By fitting a drill pad to the tailstock in place of the center and holding the drill in a chuck on the headstock, it is possible to use the lathe as a drill press for some kinds of work.

Lathes can be fitted with milling attachments for cutting keyways, grinding attachments for doing cylindrical grinding and numerous other fittings.

MILLING AND MILLING MACHINES*

Among the machine tools which should form part of the equipment of every motor repair shop, none is of greater convenience or more important than the Universal Milling Machine. Its range and adaptability to motor service shop work is suggested by the following partial list of operations which are efficiently performed on this machine: Drilling operations on yoke, fork or spring hanger; turning and cutting off pins and king bolts; milling threads on shafts and axles; boring wrist-pin holes in cylinders; boring and reaming cylinders; milling keyways and oil grooves; milling hexagon and square shafts; milling multiple splined shafts and axle ends; cutting differential bevel gears; milling timing and rotary pump gears; cutting chain sprockets, splining tapered

* See also pp. 99-100, Vol. I, Starrett Books, and pp. 81-88, Vol. II, Starrett Books.

holes by using a slotting attachment, etc., etc. A very large percentage of all the machine tool work called for in motor repair shops can be done to advantage on a universal milling machine.

The standard milling machine is of column and knee type. The knee, which carries the work table, is mounted on the column and can be adjusted vertically to bring the work into proper position for the milling cutters to operate. It may also be fed upward to take vertical cuts. The work table, mounted on the knee, is fed horizontally in two directions, one at right angles to the axis of the spindle, and the other parallel to the spindle axis. The cutterhead on one type of Universal miller that is deservedly popular with the automobile repair trade may be rotated to cut at any angle through 90° in a plane parallel to the axis of the column, and by use of a sub-head, may be swung through 270°. The sub-head, in turn, may be turned through 360° in a plane at right angles to the main cutterhead. This combination infinitely increases the variety of cuts possible with a single right angle end mill and makes a corresponding reduction in tool costs. It should be noted that while in lathe work the cutting tool is fixed and the work rotated against it, in a milling machine the work is fixed and a rotating cutter is brought in contact with it.

Milling machines are usually belt driven from countershaft, and the changes of speed obtained by belt shifting are supplemented by additional changes obtained through series of gears within the miller itself. Feeds, both longitudinal and transverse, are controlled by wheels or levers on the miller. Longitudinal feed of the table, or travel of the saddle, is in a direction at right angles to the axis of the spindle. Transverse or cross feed is movement of the saddle in or out on the knee in a direction parallel to the axis of the spindle. Vertical feed is the movement of the table and saddle vertically in a direction parallel to the axis of the column.

For purposes of illustration the Van Norman Duplex Mill-

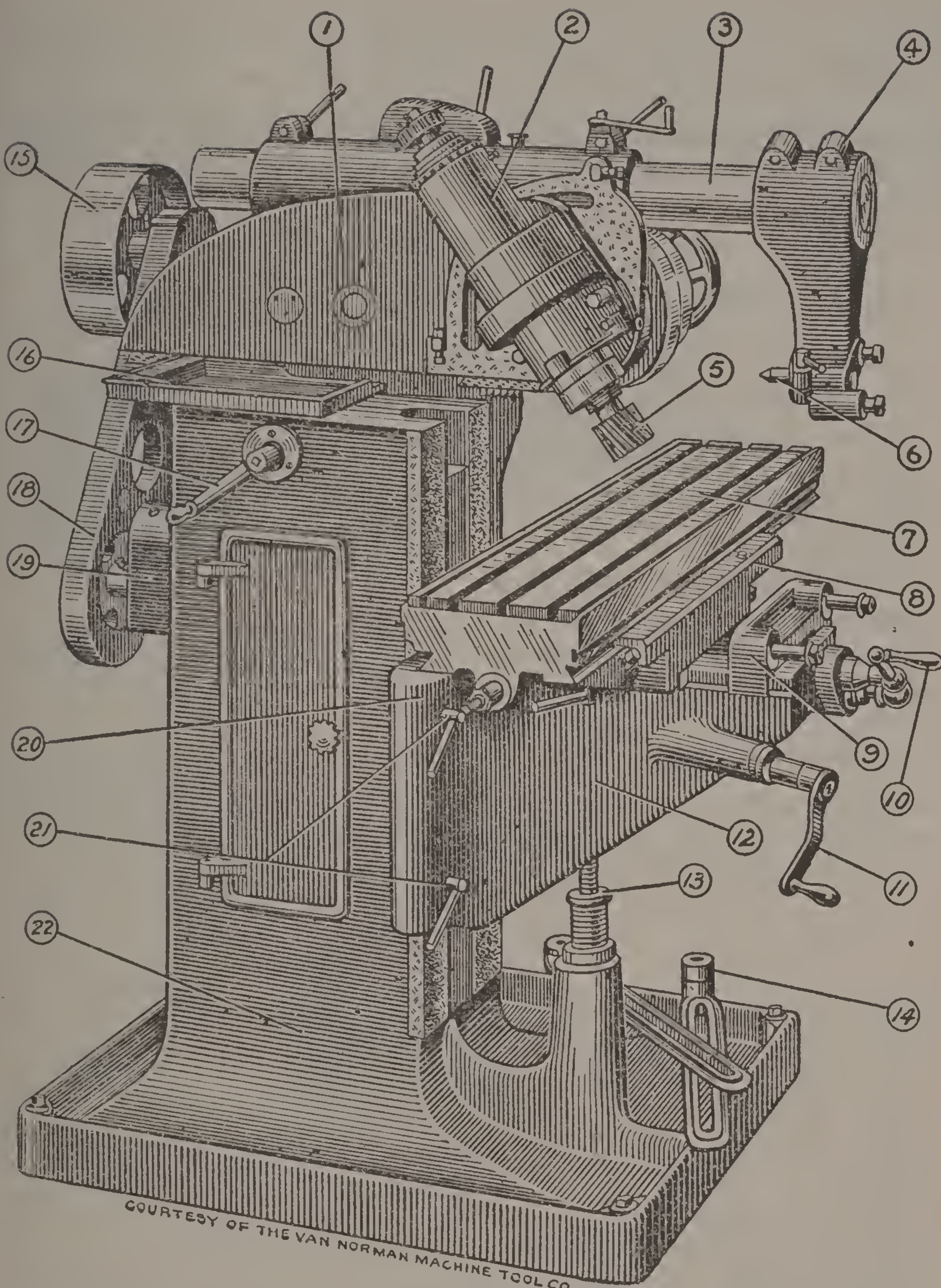


FIG. 121—DUPLEX MILLING MACHINE

1. Sliding Ram
2. Swivel Cutter Head
3. Over-Hanging Arm
4. Arm Head
5. Milling Cutter
6. Arm Head Center
7. Table
8. Saddle
9. Over-Hanging Arm Strap Block
10. Cross Feed Screw Hand Wheel
11. Knee Elevating Screw Handle
12. Knee
13. Knee Elevating Telescope Nut
14. Arm Straps
15. Main Driving Pulley
16. Tool Tray
17. Sliding Ram Handle
18. Chain Guard and Sprocket
19. Feed Box
20. Table Feed Screw
21. Knee Gib Binders
22. Column

ing Machine has been selected because it so completely meets the needs of the motor repair man. The designations of its parts are shown in Fig. 121.

MILLING KEYWAYS AND SPLINES*

In milling straight or standard keyways the vise is clamped on the milling machine table with its tongues in the bottom slot and its jaws parallel with the table ways, the shaft being held in the jaws of the vise (Fig. 122). In the case of a Ford axle shaft, which requires a keyway in its taper end, the axle must be clamped in the vise at the angle so that the top side of the taper is level horizontally. Use a surface gage in setting up the work to be sure that it lines up. The type of keyway cutter used on this job is carried in a collet in the nose of the spindle.

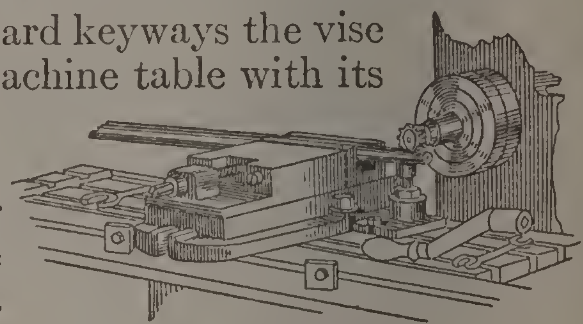


Fig. 122—Milling Straight or Standard Keyway on Tapered Shaft

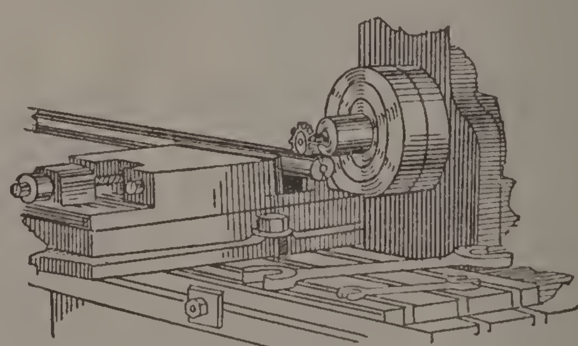
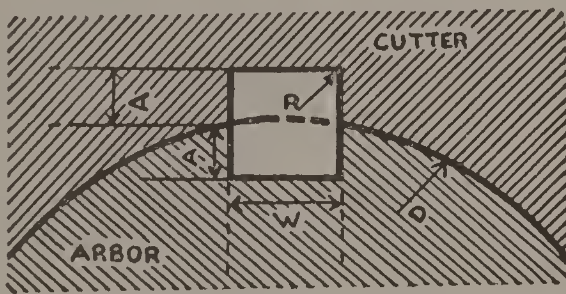


Fig. 123—Cutting Woodruff Keyway

Woodruff keyways are very common in motor replacement parts. The set-up is the same as for milling a standard keyway, but the cut is made by feeding the cutter vertically into the work. (See Fig. 123.)

* See also pp. 45-51 and p. 88, Vol. II, Starrett Books.

STANDARD KEYWAYS FOR CUTTERS AND ARBORS



Diameter of Hole (D) in Cutter, Inches	Width (W) Inches	Depth (A) Inches	Radius (R) Inches
3/8 to 9/16	3/32	3/64	.020
5/8 to 7/8	1/8	1/16	.030
5/16 to 1 1/8	5/32	5/64	.035
1 3/16 to 1 3/8	3/16	3/32	.040
*1 7/16 to 1 3/4	1/4	1/8	.050
*1 13/16 to 2	5/16	5/32	.060
2 1/16 to 2	3/8	3/16	.060
2 9/16 to 3	7/16	3/16	.060

* For all Gear Cutters of 1½ inch, 1¾ inch, 2 inch diameters, use 5/16 inch, 3/8 inch, ½ inch keys, respectively; 1½ inch uses also 3/8 inch key.

TEETH IN MILLING CUTTERS

Diameter of Cutter, Inches	Number of Teeth: Plain Roughing Cutter	Number of Teeth: Side Cutter	Number of Teeth: Plain Cutter	Radius of Point of Plain Fluting Cutter	Width of Land on Teeth
2 and 2¼	8	22	18	5/64	1/32
2½ and 2¾	8	24	18	3/32	1/32
3.................	8	24	18	7/64	1/32
3½...............	9	24	18	7/64	3/64
3¾ and 4	9	26	20	1/8	3/64
4½...............	10	26	22	1/8	3/64
5.................	10	28	22	9/64	3/64
5½...............	11	28	24	9/64	3/64
6 and 6½	12	30	24	5/32	1/16
7 and 7½	14	30	26	3/16	1/16
8.................	16	30	26	3/16	5/64
8½...............	16	32	26	3/16	5/64
9 and 9½	18	32	30	3/16	5/64
10................	20	32	30	3/16	5/64

Angle of Cutter for Side Teeth: For cutters over ¼ inch wide, 70° or 75° angle; for cutters ¼ inch wide and under, 80° angle, or 85° in extreme cases.

WOODRUFF KEY SIZES

No.	Length	Thickness	Type	Cutter Diameter
0	1/4	1/16	Half circle	1/4
00	5/16	3/32	Half circle	5/16
000	3/8	1/8	Half circle	3/8
1	1/2	1/16	Flat bottom	1/2
2	1/2	3/33	Flat bottom	1/2
3	1/2	1/8	Flat bottom	1/2
4	5/8	3/32	Flat bottom	5/8
5	5/8	1/8	Flat bottom	5/8
6	5/8	5/32	Flat bottom	5/8
61	5/8	3/16	Flat bottom	5/8
7	3/4	1/8	Flat bottom	3/4
8	3/4	5/32	Flat bottom	3/4
9	3/4	3/16	Flat bottom	3/4
91	3/4	1/4	Flat bottom	3/4
10	7/8	5/32	Flat bottom	7/8
11	7/8	3/16	Flat bottom	7/8
12	7/8	7/32	Flat bottom	7/8
A	7/8	1/4	Flat bottom	7/8
121	53/64	1/4	Flat bottom	7/8
13	1	3/16	Flat bottom	1
14	1	7/32	Flat bottom	1
15	1	1/4	Flat bottom	1
B	1	5/16	Flat bottom	1
152	1	3/8	Flat bottom	1
141	61/64	1/4	Flat bottom	1
131	15/16	5/16	Flat bottom	1
16	1 1/8	3/16	Flat bottom	1 1/8
17	1 1/8	7/33	Flat bottom	1 1/8
18	1 1/8	1/4	Flat bottom	1 1/8
C	1 1/8	5/16	Flat bottom	1 1/8
161	1	5/16	Flat bottom and ends	1 1/8
19	1 1/4	3/16	Flat bottom	1 1/4
20	1 1/4	7/32	Flat bottom	1 1/4
21	1 1/4	1/4	Flat bottom	1 1/4
D	1 1/4	5/16	Flat bottom	1 1/4
E	1 1/4	3/8	Flat bottom	1 1/4
22	1 3/8	1/4	Flat bottom	1 3/8
23	1 3/8	5/16	Flat bottom	1 3/8
F	1 3/8	3/8	Flat bottom	1 3/8
24	1 1/2	1/4	Flat bottom	1 1/2
25	1 1/2	5/16	Flat bottom	1 1/2
G	1 1/2	3/8	Flat bottom	1 1/2
126	1 3/8	3/16	Flat bottom and ends	2 1/8
127	1 3/8	1/4	Flat bottom and ends	2 1/8
128	1 3/8	5/16	Flat bottom and ends	2 1/8

WOODRUFF KEY SIZES (Continued)

No.	Length	Thickness	Type	Cutter Diameter
129	1 3/8	3/8	Flat bottom and ends	2 1/8
26	1 23/32	3/16	Flat bottom and ends	2 1/8
27	1 23/32	1/4	Flat bottom and ends	2 1/8
28	1 23/32	5/16	Flat bottom and ends	2 1/8
29	1 23/32	3/8	Flat bottom and ends	2 1/8
Rx	2	1/4	Flat bottom and ends	2 3/4
Sx	2	5/16	Flat bottom and ends	2 3/4
Tx	2	3/8	Flat bottom and ends	2 3/4
Ux	2	7/16	Flat bottom and ends	2 3/4
Vx	2	1/2	Flat bottom and ends	2 3/4
R	2 5/16	1/4	Flat bottom and ends	2 3/4
S	2 5/16	5/16	Flat bottom and ends	2 3/4
T	2 5/16	3/8	Flat bottom and ends	2 3/4
U	2 5/16	7/16	Flat bottom and ends	2 3/4
V	2 5/16	1/2	Flat bottom and ends	2 3/4
30	2 7/8	3/8	Flat bottom and ends	3 1/2
31	2 7/8	7/16	Flat bottom and ends	3 1/2
32	2 7/8	1/2	Flat bottom and ends	3 1/2
33	2 7/8	9/16	Flat bottom and ends	3 1/2
34	2 7/8	5/8	Flat bottom and ends	3 1/2
35	2 7/8	11/16	Flat bottom and ends	3 1/2
35	2 7/8	3/4	Flat bottom and ends	3 1/2

Woodruff keys are furnished in three grades: 1, special carbon steel; 2, special carbon steel heat treated; and 3, nickel steel heat treated. The nickel steel keys have a distinguishing line rolled along the top. Keys and cutters correspond in size and number with the following exceptions:

For Key No.	Use Cutter No.	For Key No.	Use Cutter No.
121	A	129	29
141	15	Rx	R
131	B	Sx	S
161	C	Tx	T
126	26	Ux	U
127	27	Vx	V
128	28		

Keys should project above the shaft one-half their thickness. To remove the key, tap it on one end until it rocks, when it can be pulled out.

Whenever possible, mill keyways with a splining cutter, as this method permits the use of heavier feeds and consequently requires less time. The set-up is the same as before, but the cutter is carried on an arbor, supported at its outer end by a pendant from the overarm. (See Fig. 124.)

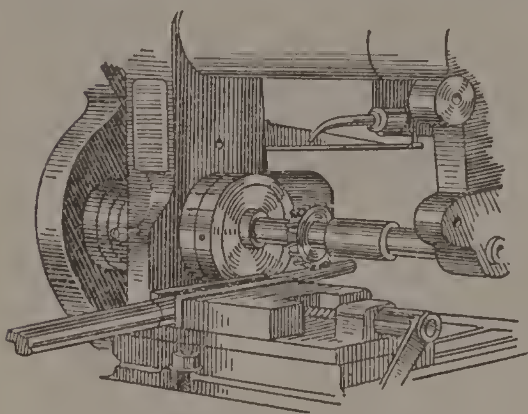


Fig. 124—Milling Keyway with Splining Cutter

For milling special keyways for which no proper cutter is to be had, a fly-cutter may be used as shown in Fig. 125. The fly-cutter consists of a lathe tool-bit ground down to the width of the keyway required and held in a fly-cutter arbor, chucked on the spindle. When using a fly-cutter the feed must be reduced to correspond to the difference between the single cutting tooth of the fly-cutter and the dozen or more cutting teeth of an ordinary milling cutter.

Certain keyways must be cut with an end mill, as shown in Fig. 126. In this operation the swivel vise is set up on the machine table and the work clamped as shown in the illustration. The vise is then swung sufficiently to align the

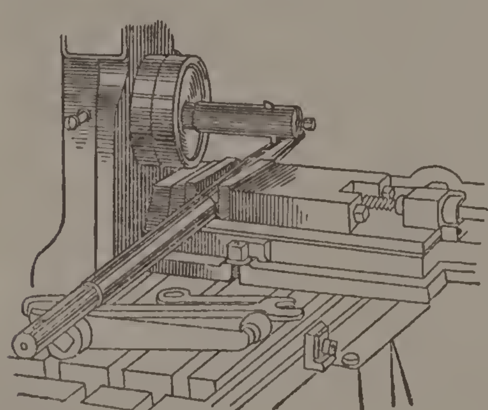


Fig. 125—Milling with Fly-Cutter

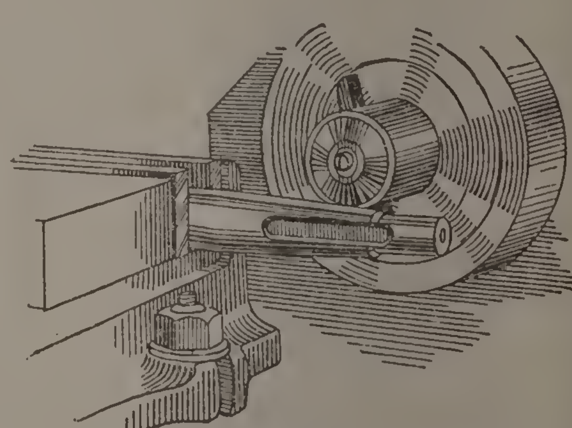


Fig. 126—Milling Keyway with End Mill

taper of the shaft with the table ways. The cutter is started at one end of the keyway and, using the horizontal feed, is allowed to travel the entire length of the keyway required. Returned to the starting point, the cutter is fed into the work a little deeper and the operation repeated.

For cutting a keyway in a taper hole a slotting attachment should be used as shown in Fig. 129. Spur gears and other parts requiring only a single keyway can be clamped to the table of the milling machine and the keyway cut with the slotting attachment set in a vertical position.

SPLINE MILLING

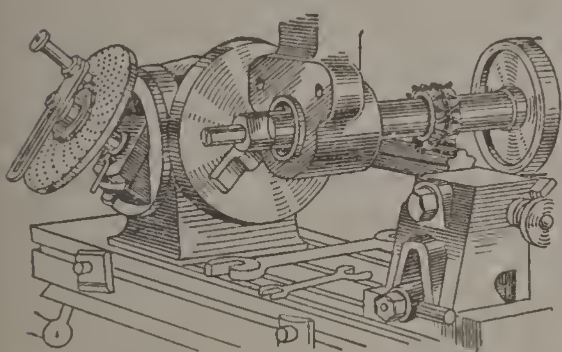


Fig. 127—Set-up for Cutting Spline

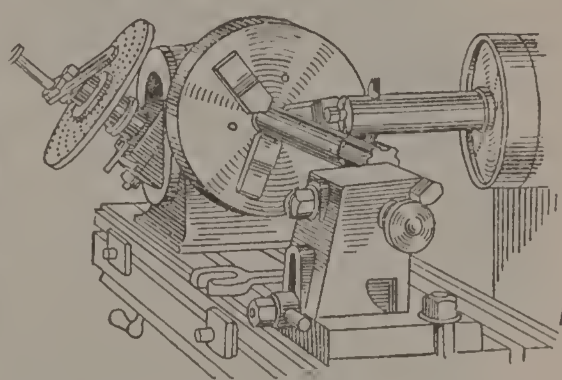


Fig. 128—Spline Milling with Fly-Cutter

Milling multiple splines calls for a very considerable degree of accuracy and skill. The operation schedule is as follows: Set a pair of milling cutters, spaced to the required key width, on the milling machine arbor, as shown in Fig. 127. This setting should be done to micrometer measurements. Then mount the work on the dividing head and tailstock centers. Set the space for the first spline exactly over the center of the shaft. To accomplish this, move the face of the cutter so that it barely touches the work. Drop the table holding the work away from the cutter. Knowing the thickness of the cutter in thousandths of an inch, move the platen holding the work, by means of the cross-feed, so that exactly one-half the diameter of the cutter is in alignment with the periphery

or surface of the work. Continue to move the platen, by means of the cross-feed, a distance exactly equal to one-half the diameter of the work. The milling cutter will then be in exact center. After setting the dividing head, index for the required number of splines and raise the work up to the cutter and depth required. Put on the longitudinal feed and mill the full length of the spline.

After the first cut withdraw the work by means of the quick traverse on the longitudinal travel and index to the next spline. Repeat until all splines are cut equally spaced and the work is ready for cutting the bottom of the splined grooves to the proper radius.

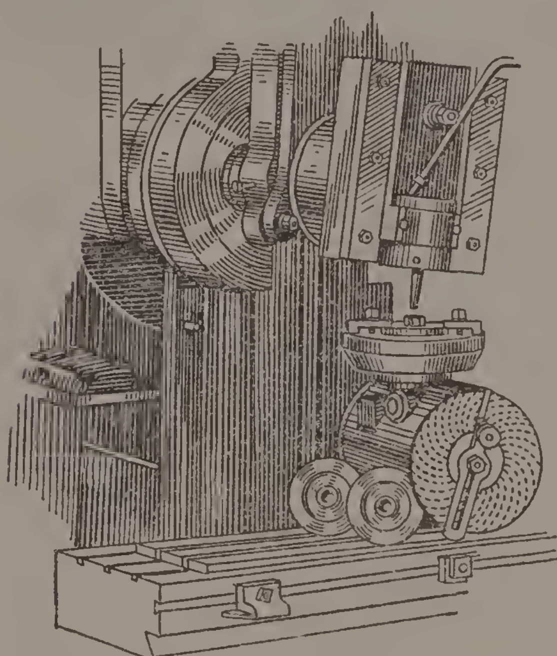


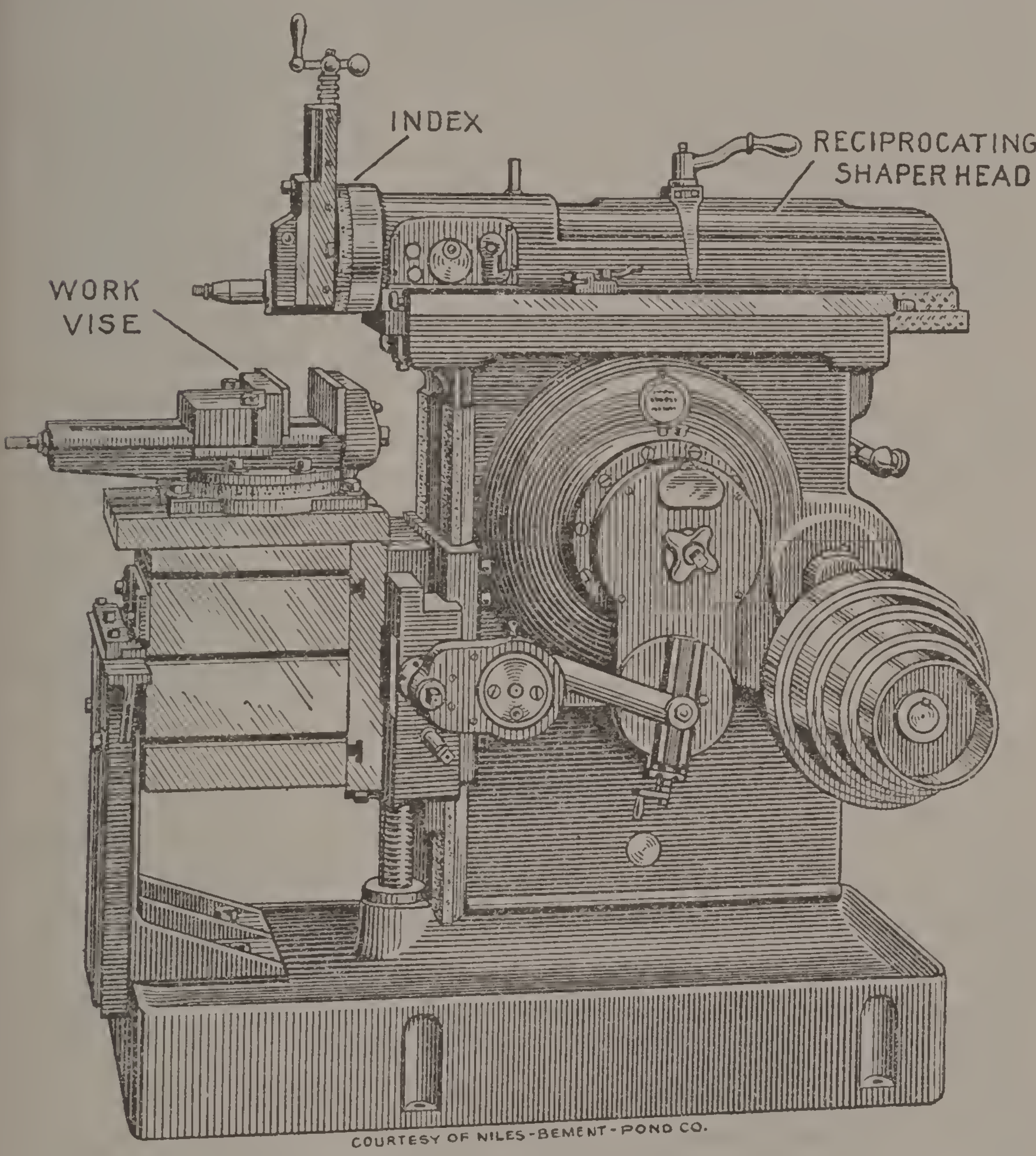
Fig. 129—Cutting Keyway in Tapered Hole

Now remove the arbor and cutter gang and push the overarm out of the way. In the spindle, insert a fly-cutter arbor carrying a tool-bit ground on the end with a radius equal to the diameter of the spline shaft. (See Fig. 128.) Set the work so that the cutter, as it rotates, will remove the metal in the splines left by the previous operation.

SHAPERS

Although many of the operations that were formerly done with shapers are now performed on milling machines, and although the latter has a much wider range of work than could possibly be done with a shaper, nevertheless, a small shaper—say one with a 16-inch stroke—will justify its being included in the equipment of any auto repair shop that seeks to handle all classes of repair work economically.

The shaper, like the milling machine, or the planer, is intended to remove metal from flat surfaces as contrasted with the round work done on a lathe or cylindrical grinder. In the shaper, the work is mounted in a fixed, work-holding vise, while the cutting tool is carried in a tool-post mounted at one end of the reciprocating shaper head. The work is moved laterally by either power or hand feed and the tool



Common Type of Shaper

holder is raised or lowered to the proper depth of cut. The tool holder or post is so mounted on an index that it may be set at any convenient angle.

GRINDING*

More and more, motor repair shops are using grinding machines in finishing operations on motor parts. Cylinder blocks, crankshafts, pistons, wrist pins, crank pins, valve stems and faces, rear axles, drive shafts, king pins, etc., are among the work now often finished on the grinding machine. Savings in time and labor as well as closer, more accurate work are responsible for the change, together with the fact that the grinding machine is more of an all-purpose tool than is any other machine tool with the possible exception of the lathe.

In the machine shop the term "grinding" refers to the producing of finished surfaces by means of rotating grinding wheels, and the process of grinding as used in finishing machine parts is today the most efficient method devised for the purpose. Grinding machines are classified into two groups, (a) those for curved surfaces; as, for example, cylindrical work; and (b) those for plane or flat surfaces. The first of these is usually called a cylindrical grinder, and the second is known as a surface grinder. Each group has many designs, made necessary by the varied uses to which grinding is adapting itself.

Grinding Wheels

These are now known as abrasive wheels, and the material from which they are made is termed an abrasive. The abrasives in common use are the minerals emery and corundum, and the manufactured abrasives, sold under the trade names of Alundum, Aloxite, Carborundum, Crystolon. Owing to the uniformity of the product as it comes from the electric furnace, manufactured abrasives are at present more largely used than natural abrasives.

* See also pp. 109-118, Vol. I, Starrett Books, and pp. 35-39, Vol. II, Starrett Books.

Abrasive Wheels

An abrasive wheel is made up of one of the above-named ABRASIVES and a BOND. The bond is, as its name indicates, something for holding the abrasive in mixture. Grinding wheels are made by three processes, known as Vitrified, Silicate and Elastic.

Vitrified Wheels

In wheels made by the Vitrified process, the bond is of earth or clay which hardens or vitrifies when subjected to a temperature of about 2500° F. to 2800° F. for a definite period of time. Various grades of hardness are obtained by using bonds of different tensile strength. The ideal bond is one which retains the grains of abrasive until sufficiently dulled by use, and then allows them to break away, bringing fresh cutting edges and points into grinding contact.

Silicate Wheels

Silicate of Sodium is the bond used in silicate wheels, and wheels made by this process are most efficient for tool and knife grinding.

Elastic Wheels

This process of bonding is generally used for the very thin wheels used for slitting metals. The principal ingredient of the bond is shellac.

Grading the Abrasive

By numerous crushing, grinding, cleansing and sorting processes, the abrasive is graded into a series of sizes which give the wheel its grain number. This number conforms to the sieve mesh through which the abrasive is passed; for example, grain No. 40 indicates that the abrasive was graded through a sieve having a mesh of forty to the linear inch.

Combination Wheels

For many grinding purposes the combination wheel is preferred to a wheel of single grade. Combination wheels are made up of abrasives of several grain numbers.

Bonding

The ideal bond is one which is impervious to moisture, does not soften by heat, and which holds firmly the cutting points of the abrasive until they become dulled by use. The bond then releases the dull abrasive and permits fresh, sharp points to begin cutting. With abrasives of equal quality the maker who nearest approaches the ideal bond produces the superior wheel.

Grading the Wheels

In grinders' language, abrasive wheels are known as hard wheels and soft wheels. The maker, therefore, lists his wheels as hard or soft by some scale of numbers or by letters. A prominent firm uses the letters of the alphabet, as shown in the following list in which "M" is medium.

Norton Grade List

The following grade list is used to designate the degree of hardness of Norton Vitrified and Silicate Wheels, both Alundum and Crystolon.

E .... Soft
F
G
H
I .... Medium Soft
J
K
L
MEDIUM .... M .... MEDIUM
N
O
P
Medium Hard .... Q
R
S
T
Hard .... U
V
W
X
Extremely Hard .... Y
Z

The intermediate letters between those designated as soft,

medium soft, etc., indicate so many degrees harder or softer; e.g., L is one grade or degree softer than medium; O, two degrees harder than medium, but not quite medium hard.

Elastic Wheels are graded as follows: 1, 1½, 2, 2½, 3, 4, 5, and 6. Grade 1 is the softest and grade 6 is the hardest.

Cylindrical Grinding

When the piece being ground is rotated, the process is known as cylindrical grinding, and the development of machines for grinding cylinders has given the process a great impetus. While it is possible to grind from the rough stock without previous lathe work, the method usually followed is to first rough turn the work.

Roughing for Grinding

This process includes the work done in removing excess stock previous to finishing to size in the grinding machine. Unless a study is made of the conditions surrounding the whole operations of the lathe and the grinding machine, lack of efficiency may result. In general, where the work is to be ground, it is best to consider the lathe as a mere roughing machine for removing the excess of stock at as deep a cut and as coarse a feed as is consistent with an efficient cutting speed, leaving the job of finishing to the grinding machine.

Amount to Leave for Grinding

If the grinding machine is modern in design, as much as 1/32 of an inch, or even more, may be left on machine steel parts for removal in the grinder, the amount varying with the size of the work itself and is largely determined by the judgment and skill of the operator. An allowance of 1/64 of an inch is general on the smaller machine parts, but this allowance should be increased on larger sizes.

Selecting the Wheel

The selection of the wheel to be used in any grinding operation can, perhaps, best be made by reference to the tables in this chapter, which fairly represent general practice. As the hardness of material and the area of contact made by the wheel have a marked influence, no table can entirely solve the problem, but it may be used as a start in the right direction. In general, a soft wheel should be used on hardened work and a harder wheel on soft materials.

In selecting a wheel for a job the following simple rules and suggestions may be of assistance:

Use wheels of the aluminous abrasive—or vitrified—type for materials of high tensile strength and hardness, such as steel. For materials of low tensile strength, such as cast iron, brass, bronze, aluminum, etc., use wheels of the silicate—or silicon carbide—type.

The more rigid the machine and the less vibration there is, the softer the wheel which may be used, and vice versa.

The narrower the line of contact between the work and the wheel, the harder the wheel must be.

While due allowance must be made for the personal factor of the operator, the condition of the grinding machine and the peculiarities of any particular job, a 36-H or 36-I Crystolon or Carborundum wheel running at approximately 4500 s.f.p.m. (surface feet per minute) will usually be found to be correct for cylinder grinding. For crankshaft grinding on bearings and pins, and for wrist pins, valve faces, rear axles, drive shafts, king pins, etc., a 6646-K Alundum wheel, running at 6000 to 6500 s.f.p.m., may be used to advantage. Where there are only a small number of valve stems to be ground, or for the casual job of this type, the same wheel may be used, but for a large number a 6646-M wheel will be found more economical. For general tool grinding in motor repair work a 6646-L or 6646-M wheel, running at about 5000 s.f.p.m., is recommended.

MOUNTING THE WHEEL

The wheel should be so mounted that there are no unequa stresses set up. Suitable guards should be provided to prevent injury to the workmen in case of the wheel bursting. Figure 130 shows RIGHT and WRONG methods of mounting wheels—carefully study the cuts.

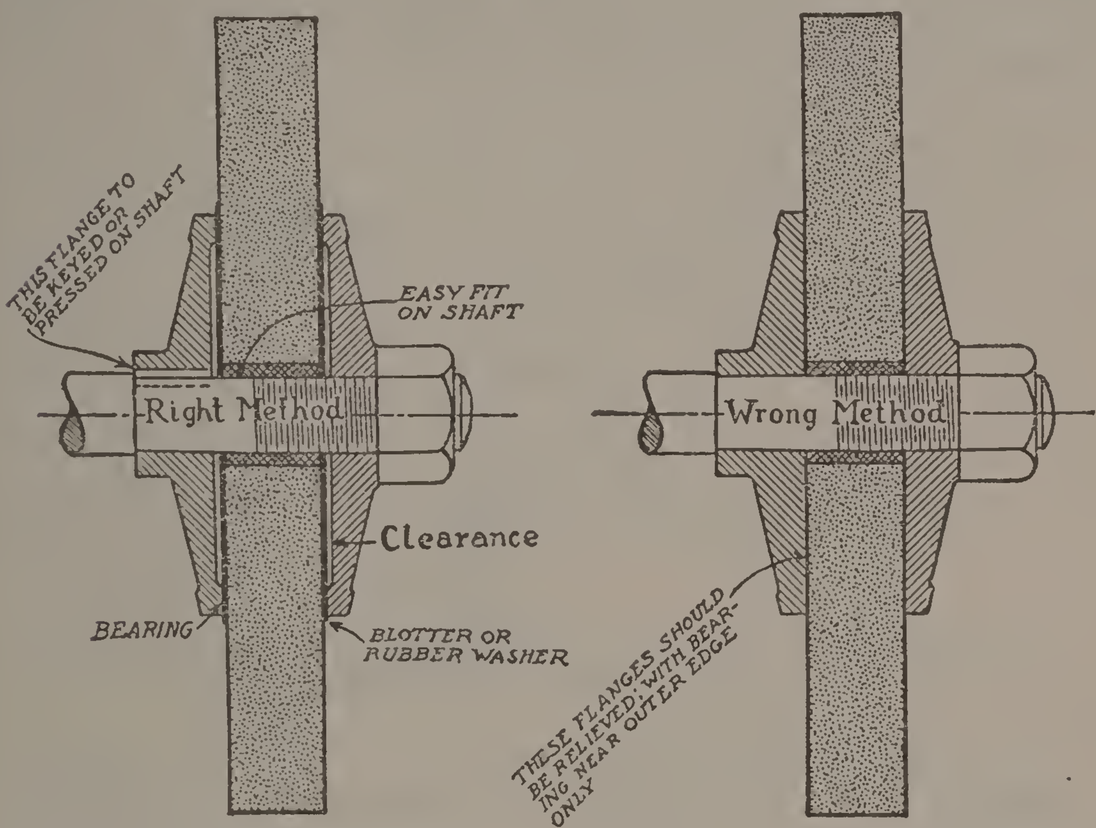


FIG. 130—RIGHT AND WRONG METHODS OF MOUNTING WHEEL

MEASURING THE WORK

The need for micrometers for obtaining exact measurements is nowhere better illustrated than in grinding. Fig. 131 shows an operator adjusting his micrometer for obtaining a measurement on a cylindrical piece. While in lathe work the position of the operator leads naturally to adjusting the micrometer spindle with the fingers of the right hand,

the left hand grasping the frame, in grinder work the reverse is generally true, hence he occupies the position as shown.

GRINDING FLAT SURFACES

Flat surface grinding may be divided into two general classes: (a) Machine parts, such as boxes, tables, cross-slides, faces of nuts, etc.; and (b) fine tool work, as, for example, steel blades, scales and rulers, straight edges, etc. Until recently the first-named class of work was done by reciprocat-

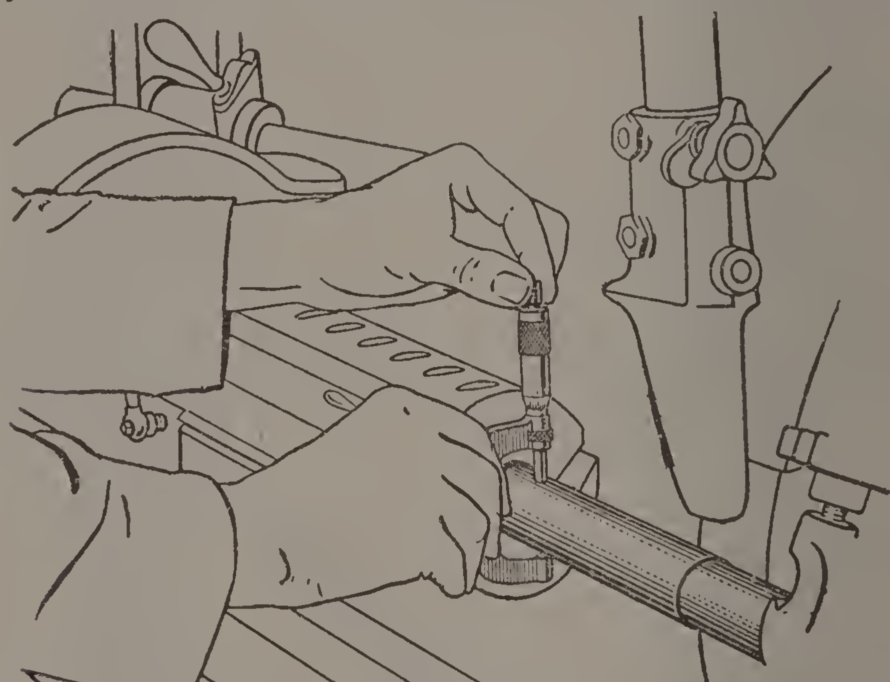


FIG. 131—USING MICROMETER ON GRINDING MACHINE WORK

ing the work beneath the circumferential face of an abrasive wheel in a machine which, in principle, is not unlike a small planer. The use of machines with CUP WHEELS has practically revolutionized such grinding, and an exactness of surface is being obtained on fine flat work which leaves little to be desired.

The operator probably has a much closer control of the cutting speed on a grinding machine than on any other machine tool because not only can the grade—or degree of hardness—of the wheel—equivalent to the cutting tool—be

changed, but also speed changes may be made in both the work and wheel revolutions. Slowing down the work revolutions, leaving the wheel turning at the same speed—produces a harder acting wheel and vice versa.

It is often claimed—and properly—that a wheel cuts better after it has been partly worn down. The reason is—kept at the same s.f.p.m.—because of the wheel's diameter, each cutting particle in its surface has more work to do and the wheel acts "softer". A soft wheel will always do more work, cut freer and use less power than a hard wheel and requires no forcing or crowding into the work to make the wheel face sharp. Another reason for a wheel's improving in cutting as it wears away, is that the arc of contact of the wheel with the work has been reduced and with the smaller diameter of the wheel the depth to which the cutting particles in the wheel are buried in the work is increased. This causes a greater strain on each cutting particle with the result that they are more quickly torn from their setting, keeping a fresh, unglazed surface in contact with the work at all times and also still further reducing the diameter of the wheel.

GRAIN NUMBERS COMMONLY USED IN VARIOUS POLISHING OPERATIONS*

Aluminum—120-150-180.
Auto Parts—24-36-60-70-80-90-120-150.
Auto Springs—36-54-70.
Bicycle Parts—70-80-90-120-150.
Brass—36-60-70-80-90-120-150.
Carriage Hardware—36-46-54.
Cast Iron—54-60-70-80-200-F-FF-FFF.
Dies—60-80-90.
Edge Tools—36-54-60-70-80-90-120-150-180.
Electric Starter Parts—24-36-60-90-120-150-180.
Forgings—36-46-54-60-70-80-90.
Glass (Beveling)—90-120-150.
German Silver—100-F.
Hammers—46-54-60-70-80-90-120-180.
High Speed Steel—36-70-80.
Knives—12-14-20-36-46-54-60-70-80-90-120.
Lapping Bushings—90-150-180-FFF.
Lapping Gages—65-F.
Lapping Machine Parts—200-FFF.
Lapping Steel Balls and Bushings—F-FFF.
Lapping Valves—FFF.
Monel Metal—F.
Machine Parts—54-60-70-100-120-150-F.
Pliers—46-54-60-70-80-90-100-120.
Renewing Files—36.
Saws—60-70-80-90-120-F-FF-FFF.
Screws—46-60-70-80-90.
Spark Plugs—80-120-F.
Tools—46-54-60-70-80-90-100-120-190.
Vises—36-46-54-60-70-80-90.
Wrenches—36-46-54-60-70-80-90-100-120.

*Courtesy of The Norton Company.

In other words, to maintain a satisfactory cutting speed slow the work revolution or increase the wheel speed if the grinding wheel acts too soft. If the wheel acts hard, increase the work revolution or decrease the wheel speed. If a wheel hums on small work, or roars a little on large work, it may be assumed that the wheel is cutting correctly. Wide faced wheels should usually be a little softer than narrow faced wheels for the same work.

WHEEL DRESSING

Truing or dressing a grinding wheel is not—or at least should not always be—done with the object of sharpening the wheel. The primary requisite for good grinding is a perfectly round, true wheel. The second requisite is a proper condition of the face of the wheel to accomplish the desired result on the work. Therefore, a wheel often should be dressed, not to "sharpen" it, but to true it up, and to put the wheel face in proper condition for the work, even though this means actually "dulling" rather than "sharpening" the wheel.

In wheel dressing great pains should be taken to avoid scoring the wheel. For this reason, a diamond tool which has been used a short time is better for truing than a new diamond, because all the little sharp corners, projections, etc., have been worn off and the diamond maintains for a longer time the same area of exposed surface presented to the wheel.

The speed with which the diamond is moved across the wheel is also a controlling factor in the work. Moving the diamond too rapidly across the wheel actually cuts a spiral or thread in the surface of the wheel and works havoc with the grinding finish. Such a wheel will invariably produce a mottled or frosted appearance on the surface of the work. The diamond should travel across the face of the wheel slowly enough to make sure that all the particles composing the surface of the wheel have been brought to the same height and are an equal distance from the center of the wheel. To

do this, take light traversing cuts with the diamond tool. Allow plenty of water to flow on the diamond while truing.

Be sure that the diamond is clamped rigidly in the tool holder on the table of the machine. The tool holder should be so set that a line running through its center and that of the diamond will strike the wheel surface at a downward angle of approximately 5° with a diameter of the wheel parallel to the plane of the table. This allows the diamond to wipe across the face of the wheel rather than dig into it, as would be done if the diamond pointed above the center of the wheel.

LAPPING

In certain lines of work the final grinding process is often made, not with abrasive wheels as previously described, but by using metal discs, rings, or cylinders, the surfaces of which have been charged with a fine flour abrasive. Such a tool is called a "lap", and its use is known as "lapping". Laps were first used by lapidaries in finishing the surfaces of mineral specimens, but laps have been in common use for a considerable time on fine work in the machine shop.

Laps are generally made of some material soft enough so that the abrasive can be readily pressed into the surface; or, as it is correctly termed, the surface is "charged". Soft, close-grained cast iron, copper, brass, or lead may be used for the lap, and any of the flour abrasives may be charged into the surface by rolling the abrasive into the lap either with a hardened roll or on a hardened surface.

In some of the finer grinding operations the lap is charged with diamond dust which has been precipitated or settled in a suitable dish of olive oil. The several grades are denoted by the time taken to precipitate; as, for example, fineness No. 5 takes ten hours.

Since lapping is a somewhat slow and tedious process it should be used for the removal of extremely small amounts.

GRINDING WHEELS FOR DIFFERENT MATERIALS

The information below is intended to give an approximate idea of the grade used under ordinary conditions.

Class of Work	Alundum		Crystolon or Carborundum	
	Grain	Grade	Grain	Grade
Aluminum castings	36 to 46	3 to 4 Elas.	20 to 24	P to R
Brass or bronze castings (large)	....	....	20 to 24	Q to R
Brass or bronze castings (small)	....	....	24 to 36	P to R
Car wheels, chilled	20	Q	16 to 24	O to S
Cast iron, cylindrical	24 comb.	J to K	30 to 46	I to L
Castings, iron (surfacing)	16 to 46	H to K	16 to 30	I to L
Castings, iron (small)	24 to 30	P to R	20 to 30	Q to S
Castings, iron (large)	16 to 20	Q to R	16 to 24	Q to S
Chilled iron castings	20 to 30	P to U	16 to 30	Q to U
Dies, chilled iron	....	....	20 to 30	O to Q
Dies, steel	36 to 60	J to L	....	....
Drop-forgings	20 to 30	P to S	....	....
Hammers, cast steel	30	P to Q	....	....
Internal cylinder grinding	....	....	30 to 60	I to L
Internal grinding, hardened steel	46 to 60	J to M	....	....
Knives (planer)	30 to 46	J to K	....	....
Lathe and planer tools	20 to 24 20 to 36	P Sil. O to P	....	....
Machine shop use, general	20 to 36	O to Q	....	....
Malleable iron castings (large)	14 to 20*	P to U*	20 to 30	Q to S
Malleable iron castings (small)	20 to 30*	P to R*	16 to 20	R to S
Milling cutters, machine grinding	46 to 60	I to M	....	....
Milling cutters, hand grinding	46 to 60	J to M	....	....
Nickel castings	20 to 24	P to Q	20 to 24	R
Pulleys, cast iron, surfacing faces of	....	....	30 to 36	K to L
Reamers, taps, etc., hand grinding	46 to 60	K to O	....	....
Reamers, taps, special machines	46 to 60	J to M	....	....
Rolls (cast iron), wet	24 to 36	J to M	24 to 46	J to M
Rolls (chilled iron), finishing	70	1½to2Elas.	70 to 80	1½to2Elas.
Rolls (chilled iron), roughing	....	....	30 to 46	2 to 5 Elas.
Saws, gumming and sharpening	30 to 50	L to N	....	....
Saws, cold cutting-off	60	O to Q	....	....
Steel (soft), cylindrical grinding	24 comb. 30 to 60	L to P L to O	....	....
Steel (soft), surface grinding	16 to 36	H to K	....	....
Steel (hardened), surface grinding	16 to 46	H to K	....	....
Steel (hardened), cylindrical grinding	24 comb. 46 to 60	K J to L	....	....
Steel, large castings	10 to 20	Q to W	....	....
Steel, small castings	20 to 30	P to R	....	....
Steel (manganese), safe work	16 to 46	L to P	....	....
Twist drills, hand grinding	46 to 60	M	....	....
Twist drills, special machines	36 to 60	K to M	....	....
Wrought iron	12 to 30	P to U	....	....
Woodworking tools	46 to 60	K to M	....	....

* Annealed.

Courtesy of The Norton Company.

GRADES OF GRINDING WHEELS

ARRANGED IN ORDER OF RELATIVE "HARDNESS" OR STRENGTH OF BOND

	Bond	Extra Soft	Very Soft	Soft	Medium Soft	Medium
Abrasive Company	Vitrified Silicate Elastic	....	G H Gs Hs ½E ¾E	I J Is Js 1E 1½E	K L Ks Ls 2E 2½E	M N Ms Ns 3E 4E
American Emery Wheel Works	Vitrified Silicate Elastic	E F	G H ½ ¾ ½E ¾E	I J 1 1½ 1E 1½E	K L 2 2½ 2E 2½E	M N 3 3½ 3E 3½E
Carborundum Company	Vitrified and Silicate Elastic	....	Z Y X 13	W V U 12 11 10	T S R P O 9 8 7	N M L K 6 5 4
Chicago Wheel and Mfg. Co.	Vitrified and Silicate	....	A A1 A2	A3 B B1	B2 B3 C C1 C2	C3 D D1 D2
Detroit Grinding Wheel Company	Vitrified and Silicate Elastic	D E	F G 1 1½	H I 2	J K 2½ 3	L M 3½ 4
Landis Tool Co.	Vitrified Silicate Carborundum	E F ¾ 1 Q P	G H 1¼ 1½ O	I J 1¾ 2 N	K L M 2¼ 2½ 3 M L K	N O 3½ 4 J
Norton Company	Vitrified and Silicate Elastic	....	....	E F 1	G H I J K 1½ 2 2½ 3	L M N O 3½ 4
Safety Emery Wheel Company	Vitrified Silicate	C C½ 6 5½	H H½ 5	A A¼ 4½	A½ A¾ M M¼ M½ 4	M¾ P P¼ P½ 3½
Sterling Grinding Wheel Company	Vitrified and Silicate Elastic	....	1 1¼ 1	1½ 1¾ 2½	2 2¼	2½ 2¾ 3
Vitrified Wheel Company	Vitrified and Silicate Elastic	....	....	C C1 ½E 1E	C2 C3 D D1 D2 1½E 2E 2½E 3E 3½E	D3 E E1 4E 4½E 5E
Waltham Grinding Wheel Company	Vitrified and Silicate Elastic	1½K 1¾K	2K 2¼K 2R	2½K 2¾K 2¼R	3K 3¼K 2½R	3½K 2¾R

Grades of Grinding Wheels—Continued

	Bond	Medium Hard	Hard	Very Hard	Extra Hard	Extremely Hard
Abrasive Company	Vitrified	O	P Q R S	U	W	Z
	Silicate	Os	Ps Qs Rs	....	....	....
	Elastic	5E	6E 7E 9*	11*	....	....
American Emery Wheel Works	Vitrified	O P	Q R	S T U	V W	Z
	Silicate	4 4½	5 6	7	....	....
	Elastic	4E 4½E	5E 6E	7E	....	....
Carborundum Company	Vitrified and Silicate	J I	H G F E	....	....	....
	Elastic	3	2 1 ½	....	....	....
Chicago Wheel and Mfg. Co.	Vitrified and Silicate	D3 E E1	E2 E3 F F1 F2	F3 G	....	G1
Detroit Grinding Wheel Company	Vitrified and Silicate	N O	P Q R	S T	....	....
	Elastic	4½	5 6	....	....	....
Landis Tool Co.	Vitrified	P	Q R S T	U	....	....
	Silicate	4½	5 5½ 6 6½	7	....	....
	Carborundum	I	II G	F U	....	....
Norton Company	Vitrified and Silicate	P Q R	S T U V	W	X	Y Z
	Elastic	5	6 7	....	....	....
Safety Emery Wheel Company	Vitrified	P¾ I I¼	I½ I¾ O O¼ O½	O¾ N N¼ N½	N¾ E E¼	E½ E¾ D
	Silicate	3	2½ 2	1½	1	....
Sterling Grinding Wheel Company	Vitrified and Silicate	3	3¼ 3½ 3¾	4 4½ 5	6	....
	Elastic	3¼	3½	....	....	....
Vitrified Wheel Company	Vitrified and Silicate	E2	E3 F F1	F2	F3	G
	Elastic	5½E	6E 6½E 7E	....	....	....
Waltham Grinding Wheel Company	Vitrified and Silicate	3¾K	4K	4¼K	4½K	4½K 5K 5½K
	Elastic	3R	3¼R	....	....	....

*Rubber Bond.

TABLE OF EQUIVALENT ANNULAR BALL BEARINGS

All bearings appearing in the same horizontal line are interchangeable and are alike in all dimensions. The table is through the courtesy of MOTOR RECORD.

Key Bearing Numbers	Hess-Bright		S. R. B. Ball	Gurney	U. S. (Strom)	Fafnir	R. I. V.	F. & S.	S. R. O.	Norma Ball	Schafer	Schatz Universal	S. K. F.	Rhineland	B. F.	New Departure	
	Regular	Monarch														Randax	S. R.
200	200	6200	200	200	200	200a	0000A	A 10	......	......	200	200	1200	200a	200a	0200	1200
201	201	6201	201	201	201	201a	000A	A 12	......	......	202B	201	1201	201a	201a	0201	1201
202	202	6202	202	202	202	202a	00A	A 15	......	......	203	202	1202	202a	202a	0202	1202
203	203	6203	203	203	203	203a	0A	A 17	......	......	203B	203	1203	203a	203a	0203	1203
204	204	6204	204	204	204	204a	1A	A 20	354b	L 20	204	204	1204	204a	204a	0204	1204
205	205	6205	205	205	205	205a	2A	A 25	355	L 25	205	205	1205	205a	205a	0205	1205
206	206	6206	206	206	206	206a	3A	A 30	356	L 30	206	206	1206	206a	206a	0206	1206
207	207	6207	207	207	207	207a	4A	A 35	357	L 35	207	207	1207	207a	207a	0207	1207
208	208	6208	208	208	208	208a	5A	A 40	358	L 40	208	208	1208	208a	208a	0208	1208
209	209	6209	209	209	209	209a	6A	A 45	359	L 45	209	209	1209	209a	209a	0209	1209
210	210	6210	210	210	210	210a	7A	A 50	360	L 50	210	210	1210	210a	210a	0210	1210
211	211	6211	211	211	211	211a	8A	A 55	361	L 55	211	211	1211	211a	211a	0211	1211
212	212	6212	212	212	212	212a	9A	A 60	362	L 60	212	212	1212	212a	212a	0212	1212
213	213	6213	213	213	213	213a	10A	A 65	363	L 65	213	213	1213	213a	213a	0213	1213
214	214	6214	214	214	214	214a	11A	A 70	364	L 70	214	214	1214	214a	214a	0214	1214
215	215	6215	215	215	215	215a	12A	A 75	365	L 75	215	215	1215	215a	215a	0215	1215
216	216	6216	216	216	216	216a	13A	A 80	366	L 80	216	216	1216	216a	216a	0216	1216
217	217	6217	217	217	217	217a	14A	A 85	367	L 85	217	217	1217	217a	217a	0217	1217
218	218	6218	218	218	218	218a	15A	A 90	368	L 90	218	218	1218	218a	218a	0218	1218
219	219	6219	219	219	219	219a	16A	A 95	369	L 95	219	219	1219	219a	219a	0219	1219
220	220	6220	220	220	220	220a	17A	A100	370	L100	220	220	1220	220a	220a	0220	1220
221	221	6221	221	221	221	221a	18A	A105	371	L105	221	221	1221	221a	221a	0221	1221
222	222	6222	222	222	222	222a	19A	A110	372	L110	222	222	1222	222a	222a	0222	1222
300	300	6300	300	300	300	300a	1B	B 10	......	M 10	302	300	1300	300a	300a	0300	1300
301	301	6301	301	301	301	301a	2B	B 12	301b	M 12	302b	301	1301	301a	301a	0301	1301
302	302	6302	302	302	302	302a	3B	B 15	302b	M 15	303	302	1302	302a	302a	0302	1302
303	303	6303	303	303	303	303a	4B	B 17	302c	M 17	303b	303	1303	303a	303a	0303	1303
304	304	6304	304	304	304	304a	5B	B 20	303	M 20	304	304	1304	304a	304a	0304	1304
305	305	6305	305	305	305	305a	6B	B 25	304	M 25	305	305	1305	305a	305a	0305	1305
306	306	6306	306	306	306	306a	7B	B 30	305	M 30	306	306	1306	306a	306a	0306	1306
307	307	6307	307	307	307	307a	8B	B 35	306	M 35	307	307	1307	307a	307a	0307	1307

Key Bearing Numbers	Hess-Bright Regular	Hess-Bright Monarch	S. R. B. Ball	Gurney	U. S. (Strom)	Fafnir	R. I. V.	F. & S.	S. R. O.	Norma Ball	Schafer	Schatz Universal	S. K. F.	Rhineland	B. F.	New Departure Randax	New Departure S. R.
308	308	6308	308	308	308	308a	9B	B 40	307	M 40	308	308	1308	308a	308a	0308	1308
309	309	6309	309	309	309	309a	10B	B 45	308	M 45	309	309	1309	309a	309a	0309	1309
310	310	6310	310	310	310	310a	11B	B 50	309	M 50	310	310	1310	310a	310a	0310	1310
311	311	6311	311	311	311	311a	12B	B 55	310	M 55	311	311	1311	311a	311a	0311	1311
312	312	6312	312	312	312	312a	13B	B 60	311	M 60	312	312	1312	312a	312a	0312	1312
313	313	6313	313	313	313	313a	14B	B 65	312	M 65	313	313	1313	313a	313a	0313	1313
314	314	6314	314	314	314	314a	15B	B 70	313	M 70	314	314	1314	314a	314a	0314	1314
315	315	6315	315	315	315	315a	16B	B 75	314	M 75	315	315	1315	315a	315a	0315	1315
316	316	6316	316	316	316	316a	17B	B 80	315	M 80	316	316	1316	316a	316a	0316	1316
317	317	6317	317	317	317	317a	18B	B 85	316	M 85	317	317	1317	317a	317a	0317	1317
318	318	6318	318	318	318	318a	19B	B 90	317	M 90	318	318	1318	318a	318a	0318	1318
319	319	6319	319	319	319	319a	20B	B 95	318	M 95	319	319	1319	319a	319a	0319	1319
320	320	6320	320	320	320	320a	21B	B100	319	M100	320	320	1320	320a	320a	0320	1320
321	321	6321	321	321	321	321a	22B	B105	320	......	321	321	1321	321a	321a	0321	1321
322	322	6322	322	322	322	322a	23B	B110	321	......	322	322	1322	322a	322a	0322	1322
403	403	6403	403	403	403	403a	1B	C 17	331	S 17	403b	403	403	403a	403a	0403	1403
404	404	6404	404	404	404	404a	2C	C 20	332	S 20	404	404	404	404a	404a	0404	1404
405	405	6405	405	405	405	405a	3C	C 25	333	S 25	405	405	405	405a	405a	0405	1405
406	406	6406	406	406	406	406a	4C	C 30	334	S 30	406	406	406	406a	406a	0406	1406
407	407	6407	407	407	407	407a	5C	C 35	335	S 35	407	407	407	407a	407a	0407	1407
408	408	6408	408	408	408	408a	6C	C 40	336	S 40	408	408	408	408a	408a	0408	1408
409	409	6409	409	409	409	409a	7C	C 45	337	S 45	409	409	409	409a	409a	0409	1409
410	410	6410	410	410	410	410a	8C	C 50	338	S 50	410	410	410	410a	410a	0410	1410
411	411	6411	411	411	411	411a	9C	C 55	339	S 55	411	411	411	411a	411a	0411	1411
412	412	6412	412	412	412	412a	10C	C 60	340	S 60	412	412	412	412a	412a	0412	1412
413	413	6413	413	413	413	413a	11C	C 65	......	S 65	413	413	413	413a	413a	0413	1413
414	414	6414	414	414	414	414a	12C	C 70	342	S 70	414	414	414	414a	414a	0414	1414
415	415	6415	415	415	415	415a	......	C 75	......	......	415	415	415	415a	......	0415	1415
416	416	6416	416	416	416	416a	13C	C 80	343	S 80	416	416	416	416a	416a	0416	1416
417	417	6417	417	417	417	417a	......	C 85	......	......	417	417	417	417a	......	0417	1417
418	418	6418	418	418	418	418a	14C	C 90	344	S 90	418	418	418	418a	418a	0418	1418
419	419	6419	419	419	419	419a	......	C 95	......	......	419	419	419	419a	......	0419	1419
420	420	6420	420	420	420	420a	15C	C100	......	S100	420	420	420	420a	420a	0420	1420

ANTI-FREEZING SOLUTIONS

All Percentages are given by volume.

DENATURED ALCOHOL SOLUTIONS

Per Cent Water	Per Cent Alcohol	Specific Gravity	Freezing Point	Boiling Point
100	0	1.000	32 degrees	212 degrees
90	10	.987	24	200
80	20	.975	13	185
70	30	.964	−2	169
60	40	.953	−20	163
50	50	.934	−34	157
40	60	.915	−47	152
30	70	.870	−57	148

WOOD ALCOHOL SOLUTIONS

Per Cent Water	Per Cent Alcohol	Specific Gravity	Freezing Point	Boiling Point
100	0	1.000	32 degrees	212 degrees
90	10	.986	20	197
80	20	.975	8	182
70	30	.962	−7	166
60	40	.949	−24	150
50	50	.934	−37	134
40	60	.917	−52	....
30	70	.850	−60	....

GLYCERINE SOLUTIONS

Per Cent Water	Per Cent Glycerine	Freezing Point	Boiling Point
100	0	32 degrees	212 degrees
90	10	28	215
80	20	23	218
70	30	14	222
60	40	6	224
50	50	0	227
40	60	−3	233

WOOD ALCOHOL AND GLYCERINE SOLUTIONS

Per Cent Water	Per Cent Glycerine	Per Cent Alcohol	Freezing Point	Boiling Point
100	0	0	32 degrees	212 degrees
90	5	5	24	204
80	10	10	15	194
70	15	15	−5	184

CALCIUM CHLORIDE SOLUTIONS*

Per Cent Water	Per Cent Calcium Chloride	Specific Gravity	Freezing Point
100	0	1.000	32 degrees
90	10	1.085	22
80	20	1.179	−1
70	30	.....	−60

* The use of Calcium Chloride solutions is not recommended.

HOW TO TELL CARBURETOR SIZES

Carbureter Size	Inside Diameter of Hole in Flange	Carbureter Size	Inside Diameter of Hole in Flange
1/2	13/16	1 1/2	1 11/16
5/8	13/16	1 3/4	1 15/16
3/4	1 1/16	2	2 3/16
7/8	1 1/16	2 1/2	2 11/16*
1	1 3/16	3	3 3/16*
1 1/4	1 7/16	3 1/2	3 11/16*

* These sizes have four bolt holes, others have two.

MAXIMUM POWER REQUIRED TO DRIVE MACHINE TOOLS

12-in. lathe	2 hp.
14-in. lathe	3 hp.
16-in. lathe	4 hp.
18-in. lathe	5 hp.
20-in. lathe	7 hp.
22-in. lathe	8 hp.
24-in. lathe	9 hp.
27-in. lathe	10 hp.
30-in. lathe	11 hp.
36-in. lathe	15 hp.
24-in. drill press	3 1/2 hp.
60-in. planer	23 hp.
42-in. planer	12 hp.
42-in. mill	19 hp.
1/2-in. sensitive drill	3/4 hp.
16-in. shaper	2 hp.
18-in. shaper	3 hp.
No. 1 Universal milling machine	2 hp.
No. 2 Universal milling machine	2 hp.
No. 3 Universal milling machine	3 hp.
6-in. grinding wheel (two wheels)	1 hp.
10-in. grinding wheel (two wheels)	2 hp.
12-in. grinding wheel (two wheels)	3 hp.
18-in. grinding wheel (two wheels)	7 1/2 hp.

WIRE AND SHEET METAL GAGES

Gage No.	American or Brown & Sharpe	Birmingham or Stubbs iron wire	Washburn & Moen iron wire	U.S. Std. for plate (iron and steel)	Stubbs steel wire	Twist drill and steel wire	Washburn & Moen music wire	Wood and m'chine screw
8–0	....	....	....	....	....	....	.0083	....
7–0	....	....	.490	.500	....	....	.0087	....
6–0	....	....	.462	.469	....	....	.0095	....
5–0	....	....	.431	.438	....	....	.010	....
4–0	.460	.454	.394	.406	....	....	.011	....
3–0	.410	.425	.363	.375	....	....	.012	.032
2–0	.365	.380	.331	.344	....	....	.013	.045
0	.325	.340	.307	.313	....	....	.014	.058
1	.289	.300	.283	.281	.227	.228	.016	.071
2	.258	.284	.263	.266	.219	.221	.017	.084
3	.229	.259	.244	.250	.212	.213	.018	.097
4	.204	.238	.225	.234	.207	.209	.019	.110
5	.182	.220	.207	.219	.204	.206	.020	.124
6	.162	.203	.192	.203	.201	.204	.022	.137
7	.144	.180	.177	.188	.199	.201	.023	.150
8	.128	.165	.162	.172	.197	.199	.024	.163
9	.114	.148	.148	.156	.194	.196	.026	.176
10	.102	.134	.135	.141	.191	.194	.027	.189
11	.091	.120	.121	.125	.188	.191	.028	.203
12	.081	.109	.106	.109	.185	.189	.030	.216
13	.072	.095	.092	.094	.182	.185	.031	.229
14	.064	.083	.080	.078	.180	.182	.033	.242
15	.057	.072	.072	.070	.178	.180	.035	.255
16	.051	.065	.053	.063	.175	.177	.036	.268
17	.045	.058	.054	.056	.172	.173	.038	.282
18	.040	.049	.048	.050	.168	.170	.040	.295
19	.036	.042	.041	.044	.164	.166	.041	.308
20	.032	.035	.035	.038	.161	.161	.043	.321
21	.028	.032	.032	.034	.157	.159	.046	.334
22	.025	.028	.029	.031	.155	.157	.048	.347
23	.023	.025	.026	.028	.153	.154	.051	.360
24	.020	.022	.023	.025	.151	.152	.055	.374
25	.018	.020	.020	.022	.148	.150	.059	.387
26	.016	.018	.018	.019	.146	.147	.063	.400
27	.0141	.016	.0173	.0171	.143	.144	.066	.413
28	.0126	.014	.0162	.0156	.139	.141	.072	.426
29	.0112	.013	.0150	.0141	.134	.136	.076	.439
30	.0100	.012	.0140	.0125	.127	.129	.180	.453
31	.0089	.010	.0132	.0109	.120	.120	....	.466
32	.0079	.009	.0128	.0101	.115	.116	....	.479
33	.0071	.008	.0118	.0093	.112	.113	....	.492
34	.0063	.007	.0104	.0085	.110	.111	....	.505
35	.0056	.005	.0095	.0078	.108	.110	....	.518
36	.0050	.004	.0090	.0070	.106	.1065	....	.532
37	.0044	....	....	.0066	.103	.1040	....	.545
38	.0039	....	....	.0062	.101	.1015	....	.558
39	.0035	....	....	....	.099	.0995	....	.571
40	.0031	....	....	....	.097	.0980	....	.584

METAL REMOVED BY DRILLING

Depth of drilling necessary to remove given weights when balancing machine parts, lightening pistons, etc. Intermediate weights may be found by adding together the depths of the holes corresponding to the different weights that go to make up the whole. The figure on the lowest line of the table is to be added in each case to the weight and the hole is measured at the side not at the center.

BRASS

Depth to Drill in Inches

Weight Oz.	One Inch Drill	Three quarter Inch Drill	One half Inch Drill	Three eighths Inch Drill	One quarter Inch Drill
50	13.25	.....	.....	.....	.....
40	10.60	18.86	.....	.....	.....
30	7.95	14.16	.....	.....	.....
20	5.30	9.44	21.22	.....	.....
10	2.65	4.72	10.60	18.86	.....
9	2.38	4.24	9.54	16.97	.....
8	2.12	3.77	8.48	15.08	.....
7	1.85	3.30	7.42	13.20	.....
6	1.59	2.83	6.36	11.31	25.46
5	1.32	2.36	5.30	9.43	21.22
4	1.06	1.88	4.24	7.54	16.96
3	.79	1.41	3.18	5.65	12.72
2	.53	.94	2.12	3.77	8.48
1	.26	.47	1.06	1.88	4.24
.9	.24	.42	.95	1.69	3.81
.8	.21	.37	.85	1.52	3.39
.7	.18	.33	.74	1.33	2.96
.6	.16	.28	.63	1.14	2.54
.5	.13	.23	.53	.95	2.12
.4	.11	.19	.43	.76	1.69
.3	.08	.14	.31	.57	1.27
.2	.05	.09	.21	.38	.85
.1	.02	.05	.11	.19	.42
Deduct for point of drill	.10	.075	.05	.04	.03

STEEL

Weight Oz.	One Inch Drill	Three quarter Inch Drill	One half Inch Drill	Three eighths Inch Drill	One quarter Inch Drill
50	14.19	25.26	.....	.....	.....
40	11.36	20.20	.....	.....	.....
30	8.52	15.16	.....	.....	.....
20	5.68	10.06	22.73	.....	.....
10	2.83	5.05	11.38	20.21	.....
9	2.55	4.55	10.23	18.18	.....
8	2.27	4.04	9.08	16.16	36.37
7	1.98	3.54	7.96	14.14	31.83
6	1.71	3.04	6.82	12.12	27.28
5	1.41	2.54	5.68	10.10	22.73
4	1.14	2.01	4.55	8.08	18.18
3	.85	1.51	3.41	6.06	13.63
2	.57	1.01	2.28	4.04	9.08
1	.28	.50	1.14	2.02	4.55
.9	.26	.45	1.02	1.83	4.08
.8	.22	.40	.91	1.62	3.64
.7	.19	.35	.79	1.40	3.17
.6	.17	.30	.67	1.22	2.72
.5	.14	.25	.57	1.01	2.28
.4	.11	.20	.45	.81	1.83
.3	.09	.15	.33	.61	1.37
.2	.06	.10	.22	.41	.91
.1	02	.05	.11	.20	.45
Deduct for point of drill	.10	.075	.05	.04	.03

CAST IRON

Weight Oz.	One Inch Drill	Three quarter Inch Drill	One half Inch Drill	Three eighths Inch Drill	One quarter Inch Drill
50	15.43	27.50	.....	.....	.....
40	12.36	22.00	.....	.....	.....
30	9.27	16.50	.....	.....	.....
20	6.19	10.94	24.75	.....	.....
10	3.08	5.47	12.38	22.02	.....
9	2.78	4.95	11.12	19.81	.....
8	2.47	4.40	9.90	17.60	39.60
7	2.16	3.86	8.66	15.41	34.65
6	1.86	3.31	7.43	13.20	29.72
5	1.54	2.75	6.18	11.00	24.75
4	1.24	2.20	4.96	8.80	19.80
3	.93	1.65	3.71	6.60	14.85
2	.62	1.09	2.48	4.40	9.90
1	.31	.55	1.24	2.20	4.96
.9	.28	.49	1.11	1.98	4.34
.8	.25	.44	.99	1.76	3.96
.7	.22	.39	.87	1.54	3.46
.6	.19	.33	.74	1.32	2.97
.5	.15	.28	.62	1.10	2.48
.4	.12	.22	.50	.88	1.98
.3	.09	.17	.37	.66	1.49
.2	.06	.11	.25	.44	.99
.1	.03	.06	.12	.22	.50
Deduct for point of drill	.10	.075	.05	.04	.03

INSTALLING FLYWHEEL RING GEARS*

Motor repair shops today are frequently called on to replace damaged flywheel, or starter, ring gears. If the gear ring is of steel, shrunk onto the cast-iron flywheel, the job is comparatively simple, it being necessary only to remove the old ring by sawing or drilling it, and shrink on a replacement ring. When, however, the gear teeth are cut in the body of the cast-iron flywheel, the work is considerably more difficult.

To replace a damaged flywheel ring gear, the teeth of which are cut in the body of the wheel, the following method should be followed: Mount the damaged flywheel in a lathe or boring mill so that it may be turned down to take the new ring. The chief difficulty in this part of the operation is centering the wheel and if the chalk method is used in truing up or centering the work, the job may well take several hours. To center the wheel quickly, place it in the chuck on a pipe center held in the tail stock of the lathe, and use an amplifying dial gage to indicate how much the wheel is running out of true. The pipe center itself centers the flywheel within a few thousandths of an inch and the work may easily be adjusted so it runs perfectly true by means of the chuck jaws.

In the turning operation the old teeth are removed, the tool being fed in from the side with a broad cut so that it works below the base of the gear teeth, avoiding vibration and chatter. When turned down to size the wheel is ready for shrinking on the ring gear.

Before shrinking the new ring gear into place it must be expanded by heating, in either a regular rim heating forge, or other suitable furnace, or by means of several blow torches if neither of the other means are available. The ring should be heated to a blue heat and care should be taken to heat all parts of it equally. When heated to the proper degree, the ring is removed from the furnace and quickly slipped over the rim of the flywheel. The contraction of the ring gear

* Data on Allowances for Shrink Fits may be found on pp. 15 and 17, Vol. II, Starrett Books.

as it cools will be sufficient to lock it firmly in place on the rim of the flywheel. To estimate the allowance for a good shrink fit the following formula may be used:

A = allowance in thousandths of an inch
D = nominal diameter of fit

$$\frac{A}{1000} = \frac{17}{16} D + .5''$$

DEFINITIONS OF ELECTRICAL TERMS

Alternating Current—(Abbrev. ac)—An electric current which alternately reverses its direction around a circuit, as distinguished from a *direct* current (dc) in which the current flows in one direction only. The process of changing an alternating current to a direct current is called *rectification* and is accomplished by an apparatus known as a *rectifier*.

Ampere—the unit of measurement of electric current. The current which is produced by the electrical pressure, or E M F, of 1 volt applied to a conductor the resistance of which is 1 ohm.

Ampere Hour—the quantity of electricity transferred by a current of 1 ampere flowing for one hour, or its equivalent; such as four amperes flowing for fifteen minutes.

Capacity—the quantity of electricity in ampere hours which may be taken from a cell or storage battery at a given rate of discharge.

Circuit—Conductors connected with a source of electrical supply are collectively called a circuit. When they form a closed path through which a current circulates there is a *closed circuit;* when there is no closed path and no current circulates, there is an open circuit.

Continuous Rating—denotes the load which the electrical machine can carry continuously without overheating or deterioration.

E M F—electromotive force—that force which causes or tends to cause an electric current.

Frequency—the number of complete periods of alteration per second of an alternating current.

Kilowatt Hour—(abbrev. kw.-hr.)—the unit commonly used in measuring electrical energy. The energy represented by 1 kw. operating for one hour or its equivalent, i. e., 3 kw. operating for 20 minutes.

Ohm—the unit of resistance. The resistance requiring an E. M. F. of 1 volt to maintain a steady current of 1 ampere.

Parallel Circuit—a circuit in which the parts of the circuit are connected in independent circuits branching from the main circuit. Also known as a shunt circuit.

Polarization—When a cell is generating current the metal dissolved from the positive plate combines with the liquid in the cell to form hydrogen gas which appears in the form of bubbles on the surface of the negative plate, diminishing the amount of surface in contact with the liquid and so increasing resistance and reducing current. The hydrogen bubbles are positive and set up an opposing E. M. F. which further reduces the current.

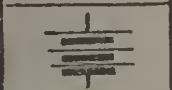
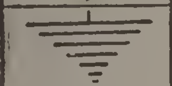
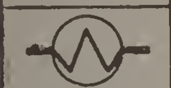
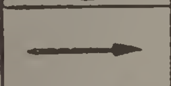
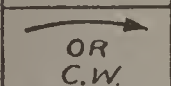
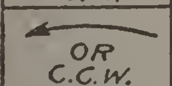
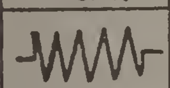
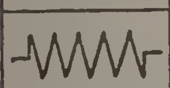
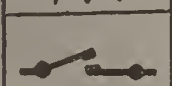
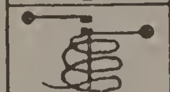
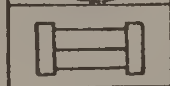
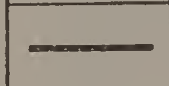
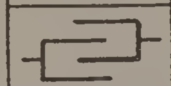
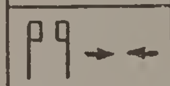
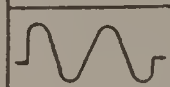
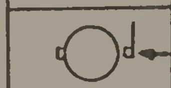
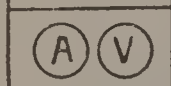
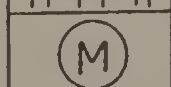
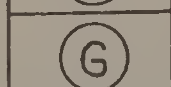
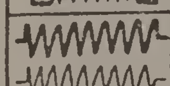
Rating—Normal Rating denotes the load which a motor or generator or transformer is designed to carry under service conditions.

Series Circuit—A circuit in which the several parts are connected so that the same current passes through all. Thus cells are in series when the positive of one is connected with the negative of another.

Volt—The unit of measurement of electromotive force. The E. M. F. required to send a steady current of 1 ampere against a resistance of 1 ohm.

Watt—The electrical unit of power. 1 horsepower equals 746 watts or .746 kilowatts.

DEFINITION OF SYMBOLS USED ON WIRING DIAGRAMS

	Positive side of battery or electrical circuit.
	Negative side of battery or electrical circuit.
	Indicates battery, either storage or dry cells.
	Indicates a grounded connection.
	Indicates a dome or instrument board light.
	Indicates direction of current flow (mostly used on internal diagrams).
	Indicates clockwise rotation.
	Indicates counter-clockwise rotation.
	Indicates a coil of heavy wire (primary circuit of ignition coil).
	Indicates a coil of light wire (secondary circuit of ignition coil).
	Indicates a shunt wound generator.
	Indicates a series wound generator.
	Indicates a connection between wires.
	Indicates crossed wires not connected.
	Indicates a rheostat or variable resistance.
	Indicates a push button or light switch.
	Indicates a starting switch.
	Indicates a three terminal motor-generator.
	Indicates a four terminal motor-generator.
	Indicates an automatic cut-out.
	Indicates location of fuse in circuit.
	Indicates heavy cables (starting circuit wiring).
	Indicates a condenser.
	Either symbol indicates the breaker points of an ignition system.
	Indicates a resistance such as a resistance unit and charging resistance.
	Indicates a movable brush and brush lifting switch.
	Indicates, "A" Ammeter and "V" voltmeter.
	Indicates a push button switch.
	Indicates a single-throw switch.
	Indicates the distributor of an ignition system.
	Indicates motor of a single unit system.
	Indicates generator of a single unit system.
	Indicates a ballast coil.
	Indicates a compound wound coil, primary circuit shown in heavy lines and secondary in light lines.

Courtesy of *American Automobile Digest.*

A TOOL FOR EVERY JOB*

To operate at a profit it is essential that the shop be well equipped. While the following items by no means constitute everything a well-found shop should have, they may be of assistance in checking up needed equipment.

HAND TOOLS

Hammers
- 1 Blacksmith's
- 2 Blacksmith's Sledge, short handle
- Blacksmith's Sledge, long handle
- 2 Machinist's Ball Pein
- 1 Machinist's Straight Pein
- 1 Rawhide Mallet
- 1 Lead or Babbitt Headed Hammer

Wrenches
- *1 Socket Wrench Set, with ratchet handle, No. 443
- 3 Stillson Wrenches, 6″, 12″, 18″
- 3 Monkey Wrenches, 6″, 12″, 18″
- 1 set Double End S Wrenches
- 1 complete set Spanner Wrenches
- 1 Bicycle Wrench, 4″
- 1 Narrow Jaw Monkey Wrench, 8″
- 2 Adjustable End Wrenches, 6″, 8″
- Set of Magneto Wrenches
- *1 Small Hand Vise, No. 200
- * Tap Wrench, No. 93-C

Pliers, etc.
- 2 Combination Pliers, 6″, 10″
- * Cut Nipper Pliers, No. 1
- Side cutting, parallel jaw Pliers
- Special anti-skid chain Pliers
- Cotter Pin Pliers
- Piston Ring Expanding Pliers
- Tinner's Snips
- Heavy Shears
- Bolt Cutter

Screw Drivers
- * Set Magneto Screw Drivers, No. 555
- 3 Screw Drivers, 6″x¼″, 10″x⅜″, 12″x½″

Files
- Bastard, 10″, round, square, and three cornered
- Second Cut, 8″, flat, half round
- Finishing Cut, 8″, flat, half round, rat tail, three cornered
- Finishing Cut, 6″, rat tail
- Set of File Handles
- File Brush
- Small and large Oil Stones

Chisels
- Cape, small, medium, large
- Chipping, small, medium, large
- RoundNose,small,medium,large
- Diamond Point, small, medium, large

Bench Equipment
- Pipe Vise
- 2 Swivel Vise, medium and large
- Surface Plate
- Bench Anvil
- * Machinist's Clamps, No. 278
- C Clamps, large, medium and small
- Angle Bender
- * Bench Block, No. 129

Miscellaneous
- Movable and Wall Benches
- Wheeled Trucks and Jacks
- Oil and Gasoline Storage Facilities
- Water Heater
- Washing Stand and Equipment
- Light Stand
- Small Blacksmith's or Brazing Forge, complete
- Anvil (500 lbs.) and Block
- Fire Extinguishing Equipment

MEASURING TOOLS

- *Machinist's Try-Squares, 6″, No. 55
- *Carpenter's Try-Squares, 24″, No. 451
- *Combination Square and Protractor, No. 9
- *Machinist's Flexible Scales, 2″, 6″, 12″, No. 320
- *Machinist's Stainless Steel Rule, 6″, No. 1000
- *Steel Tape, 50 ft., No. 512
- *Steel Shrink Rule, No. 374
- *Tempered Steel Rules, with Holder, No. 423
- *Carpenter's 2-ft. Folding Rule, No. 451
- *Spirit Level with Cross Level, No. 134
- *Micrometers
  - *Outside, 1″, 2″, 3″ and 6″, Nos. 436, 224
  - *Inside, 2″ to 8″, No. 124
  - *Screw Thread, No. 575

*Indicates Starrett Tools with Catalog Number.

Measuring Tools—Continued

*Gages
- *Taper, No. 269
- *Thread, Nos. 29, 40, 155, 473
- *Center, No. 390
- *Thickness, No. 71
- *Cylinder, No. 452
- *Vernier Height with Attachment, No. 454
- *Vernier Depth, No. 448
- *Tap and Drill, No. 185
- *Drill Point, No. 22
- *Surface, No. 257
- *Telescope, No. 229
- *Micrometer Depth, No. 440

*Calipers
- *Pocket Slide, No. 425
- *Hermaphrodite, small, medium, No. 243
- *Inside Lock Joint, small, medium, No. 37
- *Outside Lock Joint, small, medium, No. 36
- *Vernier, No. 122

Machine Tools

- Valve Grinding Machine
- Piston Grinding Machine
- Cylinder Grinder
- Lathe—preferably a Gap Lathe, and two or more in the following sizes: 10″, 14″, 18″, 24″. If but one is installed, it should be the 24″ size.
- Shaper, 16″ stroke
- Universal Miller
- Bench Drill
- Sensitive Drill Press
- Vertical Drill Press, heavy
- Radial Drill Press
- Arbor Press
- Hack Saw Machine

Miscellaneous Tools

- Service Car, equipped with wrecking crane, chain hoists and own set of hand tools
- Portable Shop Crane
- Wheel Puller
- Gear Puller
- Breast Drill, 2 speeds
- Electric Drill
- Gasoline Blow Torch
- Set of Number Drills
- 2 Sets of Taps and Dies, S. A. E. Standard, and U. S. Standard
- 2 Jack Screws
- Bending Bars
- Pneumatic Hammer and Riveter
- Air Compressor for Tool Operation
- Welding Table
- Oxy-Acetylene Welding Torch and Tanks complete with Welding Tools and Rods
- Carbon Burning Outfit
- Radiator Work Bench, Tank and Tools
- Overhead Trolley Tracks fitted with chain hoists
- Auto-Turn
- Motor and Rear Axle Stand
- Piston Aligner

Miscellaneous Hand Tools

- Valve Spring Lifters
- Valve Seat Reamers
- Complete set Critchley Type Expansion Reamers, 3/16″ to 1 15/16″
- Complete set Critchley Type Aligning Reamers with Pilots
- Hand Drill, 1 speed
- Belt Punch
- Bit Brace with Ratchet and complete set of Bits
- Carpenter's Cross-Cut Saw
- *2 Hacksaw Frames No. 169-144 and Blades (See Starrett Hacksaw Chart)
- *Slotting Saw, No. 249
- *Ratchet Drill Set, No. 443
- Spring Winder
- Chassis Spring Spreader
- Carbon Scrapers
- Bearing Scrapers
- Jack Plane
- Wood Chisels, 1/4″ to 2″
- *Center Punches, No. 264
- *Drive Pin Punches, 1/16″ to 5/16″ point, Nos. 565 and 248
- Soldering Irons
- *Spring Dividers, small, medium, No. 277
- *Universal Dial Test Indicator, No. 196
- *Hand Vise, No. 86
- *Decimal Equivalent Tables, Nos. 589, 590, 591
- *Tap and Drill Gage Tables, No. 185
- *Drill Blocks, No. 271
- *Toolmakers' Steel Clamps, Nos. 161, 160
- *Bevel Protractor, No. 490
- *Protractor, No. 364
- *Center Tester, No. 65
- *Set of Scribers, No. 67

* Indicates Starrett Tools with Catalog Number.

ASSORTMENTS OF DRILLS FOR TAPPING

Assortments include both tap and body drills.

No. 1—For National Fine and Coarse Threads,* No. 0 to 1 inch, 50 drills:
2, 3, 8, 11, 15, 17, 19, 21, 26, 28, 29, 30, 33, 36, 38, 39, 42, 44, 46, 49, 50, 51, 52, 53, 56, 1/4, 17/64, 5/16, 21/64, 23/64, 3/8, 25/64, 27/64, 7/16, 29/64, 31/64, 1/2, 33/64, 17/32, 9/16, 37/64, 5/8, 21/32, 11/16, 3/4, 49/64, 7/8, 15/16, 1.

No. 2—For National Coarse Threads, No. 1 to 1 inch, 23 drills:
2, 8, 11, 17, 19, 26, 28, 29, 30, 33, 36, 39, 44, 47, 49, 51, 53, 1/4, 5/16, 23/64, 3/8, 27/64, 7/16, 31/64, 1/2, 17/32, 9/16, 5/8, 21/32, 3/4, 49/64, 7/8, 1.

No. 3—For National Fine Threads, No. 0 to 1 inch, 39 drills:
2, 3, 11, 15, 19, 21, 28, 29, 30, 33, 38, 39, 42, 44, 46, 49, 50, 52, 53, 56, 1/4, 17/64, 5/16, 21/64, 3/8, 25/64, 7/16, 29/64, 1/2, 33/64, 9/16, 37/64, 5/8, 11/16, 3/4, 13/16, 7/8, 15/16, 1.

No. 4—For National Fine and Coarse Threads, No. 6 to 3/4 inch, 33 drills:
2, 3, 8, 11, 15, 17, 19, 21, 26, 28, 29, 30, 33, 1/4, 17/64, 5/16, 21/64, 23/64, 3/8, 25/64, 27/64, 7/16, 29/64, 31/64, 1/2, 33/64, 17/32, 9/16, 37/64, 5/8, 21/32, 11/16, 3/4.

No. 5—For National Fine and Coarse Threads, No. 6 to 9/16 inch, 27 drills:
2, 3, 8, 11, 15, 17, 19, 21, 26, 28, 29, 33, 36, 1/4, 17/64, 5/16, 21/64, 23/64, 3/8, 25/64, 27/64, 7/16, 29/64, 31/64, 1/2, 33/64, 9/16.

No. 6—For National Coarse Threads, No. 6 to 9/16 inch, 18 drills:
2, 8, 11, 17, 19, 26, 28, 29, 36, 1/4, 5/16, 23/64, 3/8, 27/64, 7/16, 31/64, 1/2, 9/16.

No. 7—For National Fine Threads, No. 6 to 9/16 inch, 20 drills:
2, 3, 11, 15, 19, 21, 28, 29, 33, 1/4, 17/64, 5/16, 21/64, 3/8, 25/64, 7/16, 29/64, 1/2, 33/64, 9/16.

No. 8—For Pipe Threads, 1/8 inch to 3/4 inch, 5 drills:
21/64, 27/64, 9/16, 11/16, 29/32.

No. 9—For Standard Taper Pins, No. 00 to No. 6, 8 drills:
32, 29, 27, 21, 15, 4, 1/4, 9/32.

No. 10—For Cotter Pins, 1/16 to 13/64 inch, 7 drills:
2, 11, 21, 28, 30, 36, 48.

No. 11—For Wood Screws, No. 4 to No. 18, 38 drills:
2, 3, 4, 6, 8, 12, 14, 16, 17, 19, 20, 23, 24, 25, 26, 28, 30, 31, 32, 33, 35, 38, 40, 42, 44, 45, 48, 49, 51, 52, 15/64, 1/4, 17/64, 9/32, 19/64.

No. 12—For General Automobile Shop use. Includes all body and tap drills necessary for National Fine and Coarse Threads, No. 6 to 9/16 inch, Pipe Threads 1/8 to 3/8 inch, Standard Taper Pins, No. 00 to No. 6, Cotter Pins 1/16 to 13/64 inch, Wood Screws, No. 4 to No. 18, 50 drills:
2, 3, 4, 8, 11, 12, 14, 15, 16, 17, 19, 20, 21, 23, 24, 25, 26, 27, 28, 29, 30, 31, 32, 33, 36, 38, 40, 42, 44, 45, 48, 49, 51, 52, 1/4, 17/64, 9/32, 5/16, 19/64, 21/64, 23/64, 3/8, 25/64, 27/64, 7/16, 29/64, 31/64, 1/2, 33/64, 9/16.

*See pages 85, 97 and 98.

CHAIN DISCOUNTS

Many automobile accessories, etc., are sold to the trade subject to what are known as chain discounts and which frequently are not only difficult to figure, but actually misleading. The following table is, therefore, submitted. To use it, multiply the list price by the figure in the extreme right-hand column that corresponds to the chain discount given. For example: A given article or part costs $12.00 less a chain discount of 25%+ 10%+ 5%. This discount is found in the seventh line from the top of the table, and, according to the figures under "Total Discount", is equivalent to .35875. To find the net cost of the goods, multiply the list price by 100 less .35875 or . 64125, the figure given in the right-hand column.

Chain Discounts	Total Disc.	Multiply by
25% =	.25	.75
25% + 5% =	.2875	.7125
25% + 5% + 5% =	.32312	.67688
25% + 5% + 5% + 5% =	.35697	.64303
25% + 10% =	.325	.675
25% + 10% + 10% =	.3925	.6075
25% + 10% + 5% =	.35875	.64125
25% + 10% + 5% + 5% =	.39081	.60919
25% + 10% + 5% + 5% + 5% =	.42127	.57873
20% =	.20	.80
20% + 5% =	.24	.76
20% + 5% + 5% =	.278	.722
20% + 5% + 5% + 5% =	.3141	.6859
20% + 10% =	.28	.72
20% + 10% + 10% =	.352	.648
20% + 10% + 5% =	.316	.684
20% + 10% + 5% + 5% =	.3502	.6498
20% + 10% + 5% + 5% + 5% =	.38269	.61731
15% =	.15	.85
15% + 5% =	.1925	.8075
15% + 5% + 5% =	.23287	.76713
15% + 5% + 5% + 5% =	.27123	.72877
15% + 10% =	.235	.765

Chain Discounts—Continued

Chain Discounts	Total Disc.	Multiply by
15% + 10% + 10% =	.3115	.6885
15% + 10% + 5% =	.27325	.72675
15% + 10% + 5% + 5% =	.30959	.69041
15% + 10% + 5% + 5% + 5% =	.34411	.65589
10% =	.10	.90
10% + 10% =	.19	.81
10% + 5% =	.145	.855
10% + 5% + 5% =	.18775	.81225
10% + 5% + 5% + 5% =	.22836	.77164
5% =	.05	.95
5% + 5% =	.0975	.9025
5% + 5% + 5% =	.14262	.85738
5% + 5% + 5% + 5% =	.18549	.81451
33⅓% =	.3333	.6667
33⅓% + 5% =	.3667	.6333
33⅓% + 5% + 5% =	.3984	.6016
33⅓% + 5% + 5% + 5% =	.4283	.5717
33⅓% + 10% =	.40	.60
33⅓% + 10% + 10% =	.46	.54
33⅓% + 10% + 5% =	.43	.57
33⅓% + 10% + 5% + 5% =	.4585	.5415
33⅓% + 10% + 5% + 5% + 5% =	.48557	.51443
50% =	.50	.50
50% + 5% =	.525	.475
50% + 5% + 5% =	.54875	.45125
50% + 5% + 5% + 5% =	.57131	.42869
50% + 10% =	.55	.45
50% + 10% + 10% =	.595	.405
50% + 10% + 5% =	.5725	.4275
50% + 10% + 5% + 5% =	.59387	.40613
50% + 10% + 5% + 5% + 5% =	.61418	.38582
60% =	.60	.40
60% + 5% =	.62	.38
60% + 10 % =	.64	.36
60% + 10% + 5% =	.658	.342
70% =	.70	.30
70% + 5% =	.715	.285
70% + 10% =	.73	.27
70% + 10% + 5% =	.7435	.2565

INDEX

THE STARRETT BOOK

FOR MACHINISTS' APPRENTICES

PRICE 75 CENTS

Digest of Contents of Vol. I of The Starrett Books

Digest of Contents of Vol. I of The Starrett Books

THE STARRETT DATA BOOK

PRICE 75 CENTS

Digest of Contents of Volume II of The Starrett Books

Digest of Contents of Vol. II of The Starrett Books

Digest of Contents of Vol. II of The Starrett Books